UNIVERSITY OF CINCINNATI

Date: 11/30/2007

I, Istvan Szabo,
hereby submit this work as part of the requirements for the degree of:
Doctor of Philosophy (Ph.D.)

in:
Aerospace Engineering and Engineering Mechanics

It is entitled:

A Numerical Study of Water Injection on Transonic
Compressor Rotor Performance

This work and its defense approved by:

Chair: Dr. Mark G. Turner
Joseph P. Veres
Dr. Paul D. Orkwis
Dr. Awaterf Hamed

A Numerical Study of Water Injection on Transonic Compressor Rotor Performance

A dissertation submitted to the Division of Research and Advanced Studies
of the University of Cincinnati

in partial fulfillment of the requirements for the degree of

**DOCTORATE OF PHILOSOPHY
(Ph.D.)**

**in the Department of Aerospace Engineering and
Engineering Mechanics
of the College of Engineering**

**2007
by
Istvan Szabo**

Committee Chair: Dr. Mark G. Turner

ABSTRACT

In this study, numerical simulations of two-phase flow in a transonic compressor rotor (NASA rotor 37) were performed. Both flow and droplets' governing equations were formulated and solved in the reference frame of the rotating blades. An Eulerian-Lagrangian approach was used for the continuous and discrete phases with a two-way interaction model to simulate the mass-, momentum- and energy exchange between the different phases. Water particles were injected at the inlet with uniform particle mass flux, fully evaporating inside the rotating blade row. The phase change was most intense in areas adjacent to shock waves, where the slip velocity of the droplets was the highest. Results show decreased circumferentially averaged total temperature ratio of the air-vapor mixture across the span, which is the direct result of inter-phase energy coupling. An entropy based approach to calculate the isentropic efficiency of a wet compression process in a transonic compressor rotor was also presented. Under the proposed method, the viscous dissipation function was calculated everywhere in the domain in the post-processing phase of the numerical simulation and integrated to the wall, with special treatment in the near-wall regions where high rates of entropy generation occur. For a water to air mass flow ratio of 1% results show increased entropy production across the span, resulting in a 5% drop in compressor isentropic efficiency. Analytical integration of wall functions and numerical integration of the viscous dissipation function allows for reasonable results even with relatively coarse grids and can also be applied for single-phase flows. A parametric study of the effect of initial particle parameters on the wet compression process was also performed. Several speedlines have been computed with different amounts of water, especially near the tip. Results show that numerical stall can be delayed with injection of water near the tip, due to the increase of the axial momentum of the fluid in the endwall region, which is the direct result of phase change.

ACKNOWLEDGEMENT

I would like to thank the members of my dissertation committee, Dr. Mark G. Turner, Professor Awatef Hamed, Dr. Paul D. Orkwis and Joseph P. Veres for their valuable advices and inputs in the course of this research. It would not have been possible to complete this work without the help of Robert Ogden, who provided excellent support with hardware and software at any time of the day and week, even on weekends.

I would also like to thank my wife, Eliana C. Corredor, for her continuous support and love during the course of this work.

NOMENCLATURE

A	Area
A_j	Droplet Surface Area
α_d	Discrete Phase Volume Fraction
α_c	Continuous Phase Volume Fraction
c_μ	k- ϵ Turbulence Model Constant
c_p	Specific Heat on Constant Pressure
$\vec{u}_p$	Particle Velocity Vector
c_{diss}	Dimensionless Dissipation Coefficient
D_p	Particle Diameter
D_v	Binary Diffusion Coefficient
D_0	Initial Particle Diameter
e	Specific Internal Energy
ϵ	Eddy Dissipation
H	Enthalpy
h	Specific Enthalpy
h_{fg}	Latent Heat of Evaporation
h_j	Convective Heat Transfer Coefficient
$h_{s,j}$	Enthalpy of Mixture at Droplet-Fluid Interface
k	Conductive Heat Transfer Coefficient

k_{eff}	Effective Conductive Heat Transfer Coefficient
M	Torque acting on the Rotor, Universal Gas Constant
m_p	Particle Mass
Nu_D	Particle Nusselt Number
$\vec{u}$	Flow Velocity Vector
Φ	Viscous Dissipation Function
μ	Molecular Viscosity
μ_t	Turbulent Viscosity
μ_{eff}	Effective Viscosity
V	Volume
$\vec{n}$	Surface Normal Vector
p	Pressure
Pr	Prandtl Number
R	Gas Constant
T	Temperature
τ_{ik}	Viscous Stress Tensor in Tensor Notation
τ_T	Particle Thermal Response Time
τ_v	Particle Response Time
τ_m	Particle Evaporation Time
t	Time
T_p	Particle Temperature

$u \quad v \quad w$	x, y and z Velocity Components of the Fluid, Respectively
u^+	Streamwise Velocity in Wall Units
ε	Eddy Dissipation
ρ	Density
$\eta_c \quad \eta_t$	Compressor and Turbine Isentropic Efficiencies
ν	Kinematic Viscosity
s	Specific Entropy
$\dot{S}_{gen}$	Entropy Generation Rate
Sc	Schmidt Number
Sh_D	Particle Sherwood Number
$\vec{\nabla}$	Lagrangian Operator
δ_{ik}	Kronecker-Delta Function
y^+	Distance from the Wall in Wall Units
y_{wall}^+	Distance of the Center of the Wall-Adjacent Cell
w	Vapor Mass Fraction
W	Steam to Air Ratio
w_c	Compressor Specific Work
$\dot{Q}$	Rate of Heat Flow from Surroundings to System
$\dot{q}_j$	Rate of Convective Heat Transfer to Particle j
λ	Evaporation Constant
$\omega_{V,\infty}$	Mass Concentration of Water Vapor at Free Stream

$\omega_{V,r=R_0}$

Mass Concentration of Vapor at Particle Surface

Subscripts:

$1, 2$

Compressor Inlet and Exit

c, d

Bulk phase, Discrete Phase

p

Particle

AVG

Average across Boundary Layer

s, t

Isentropic, Total Property

Over-symbols:

—

Time-Average

'

Instantaneous

-

Vector

TABLE OF CONTENTS

ABSTRACT	iii
ACKNOWLEDGEMENT	v
NOMENCLATURE	vi
LIST OF FIGURES	xiii
LIST OF TABLES	xix
CHAPTER 1. INTRODUCTION	1
1.1. Gas Turbine Evaporative Cooling Basics.....	1
1.2. Previously Published Work.....	8
1.2.1. Effects of Water Injection on Gas Turbine Operation	8
1.2.2. Efficiency of Wet Compression	14
1.2.3. Active Compressor Stall Management	15
1.3. Objectives and Scope of the Current Study.....	16
CHAPTER 2. METHODOLOGY	20
2.1. Governing Equations	20
2.1.1. Equations of the Discrete Phase	20
2.1.1.1. Response Time	23
2.1.1.2. Stokes Number	26
2.1.1.3. Mass and Energy Transfer through Evaporation	28
2.1.2. Equations of the Mixture.....	33
2.1.3. Turbulence Model	40
2.2. Computational Model	42

2.2.1. Discretization of Governing Equations.....	42
2.2.2. Wall Function Approach.....	44
2.2.3. Boundary and Initial Conditions	47
2.2.4. Grids	48
2.2.5. Numerical Methods	50
2.2.5.1. Solution Procedure for the Equations of the Continuous Phase	50
2.2.5.2. Solution Procedure for the Equations of the Discrete Phase	52
2.2.5.3. Two-way Inter-Phase Coupling Method	53
2.3. Computational Model Validation	60
CHAPTER 3. RESULTS AND ANALYSES.....	61
3.1. Effect of Water Injection on Rotor Performance under Design Conditions	61
3.1.1. Uniform Inflow Injection, 1% Water to Air Mass Flow Ratio at Inlet	63
3.1.2. Injection near the Tip with 0.5% Water to Air Mass Flow Ratio at Inlet	74
3.1.3. Comparison of Compressor Specific Work for Wet and Dry Cases	78
3.2. An Entropy Based Definition of Wet Compression Efficiency.....	80
3.2.1. Background	80
3.2.2. The Second Law of Thermodynamics	82
3.2.3. Derivation of the Entropy Transport Equation for the Mixture.....	85
3.2.4. The Viscous Dissipation Function	89
3.2.5. Reynolds Averaged Entropy Transport Equation of the Mixture	90
3.2.6. Entropy Generation in Turbulent Shear Layers	93
3.2.7. Wall function treatment.....	96
3.2.8. Entropy Generation Application for Wet Compression	102

3.2.9. <i>Integrated Results of Entropy Generation</i>	113
3.2.10. <i>Entropy generation of the continuous phase due to inter-phase coupling</i>	122
3.2.11. <i>Entropy Production of Multiple Phases due to Inter-Phase Coupling</i>	123
3.2.12. <i>Accuracy Assessment</i>	125
3.3. <i>Parametric Study of Water Injection on Rotor Performance</i>	126
3.3.1. <i>Effect of Initial Droplet Diameter</i>	126
3.3.2. <i>Effect of the Injection of a Distribution of Particles</i>	134
3.3.3. <i>Effect of Initial Droplet Temperature</i>	139
3.3.4. <i>Effect of Water to Air Mass Flow</i>	141
3.4. <i>Off-design Performance with Water Injection</i>	145
3.4.1. <i>Shock Position for the Dry and Wet 0.5% Cases</i>	146
3.4.2. <i>Change in the Axial Momentum of Tip Flows due to Water Injection</i>	148
3.4.3. <i>Computed Speedlines for Dry and Wet Compression Cases</i>	155
CHAPTER 4. CONCLUSIONS AND FUTURE WORK	161
LIST OF REFERENCES	164
APPENDIX A: Multi-Phase Model Validation	176
A.1. <i>Grid Independence Study</i>	178
A.2. <i>Behavior of the Two-Phase Model</i>	180
APPENDIX B: Effect of Initial Particle Temperature on Inter-Phase Energy Exchange	183
APPENDIX C: Fluent Solution Settings	184
APPENDIX D: Grid File Format Conversion	186
APPENDIX E: Circumferential Averaging	187

LIST OF FIGURES

Figure 1.1. Evaporative Cooling Process on Psychometric Chart.	7
Figure 1.2. Gas Turbine Installation with Inlet Cooling (courtesy of Mee Industries Inc.).	7
Figure 2.1. Variation of Sphere Drag Coefficient with Slip Reynolds Number.	23
Figure 2.2. Variation of Momentum and Thermal Response Time with Droplet Diameter.	25
Figure 2.3. Control Volume Enclosing Part of the Mixture.	35
Figure 2.4. Illustration of Computational Cells on the Domain of Interest.	43
Figure 2.5. NASA Rotor 37 Computational Grid.	49
Figure 2.6. Meridional View of NASA Rotor 37 with Aerodynamic Survey Locations.	50
Figure 2.7. Computational Cell with Particle Trajectories.	55
Figure 2.8. Schematic Illustration of the Calculation Procedure of the Eulerian-Lagrangian Solver.	59
Figure 2.9. Residual Convergence History of a Typical Case.	59
Figure 3.1. Particle Trajectories in Rotating Blade Frame of Reference, color represents Particle Diameter.	64
Figure 3.2. Particle Trajectories colored by Tracking Time Step, $L=0.001\text{m}$	66
Figure 3.3. Particle Trajectories colored by Tracking Time Step, $L=0.0001\text{m}$	66
Figure 3.4. Relative Mach Number Contours at 50% Span for Dry Case.	68
Figure 3.5. Relative Mach Number Contours at 50% Span for Wet Case , $W=1\%$	68
Figure 3.6. Relative Mach Number Contours at Tip for Dry Case.	69
Figure 3.7. Relative Mach Number Contours at Tip for Wet Case, $W=1\%$	69

Figure 3.8. Comparison of Circumferentially Averaged Total Pressure Ratios for Wet 1% and Dry Compression.	71
Figure 3.9 Comparison of Circumferentially Averaged Total Temperature Ratios for Wet 1% and Dry Compression.	71
Figure 3.10. Particle Traces in Rotating Frame colored by Relative Flow Velocity Contours.	73
Figure 3.11. Particle Traces in Rotating Frame Colored by Droplet Slip Reynolds Number Contours.....	73
Figure 3.12. Particle Traces in Relative Reference Frame Colored by Droplet Mass [kg], Tip Injection Case, W=0.5%.	75
Figure 3.13. Particle Trajectories Colored by Residence Time [sec], Tip Injection Case, W=0.5%.	75
Figure 3.14. Comparison of Circumferentially Averaged Total Pressure Ratios for Wet and Dry Compression, Tip Injection Case, W=0.5%.	77
Figure 3.15. Comparison of Circumferentially Averaged Total Temperature Ratios for Wet and Dry Compression, Tip Injection Case, W=0.5%.	77
Figure 3.16. Schematic Representation of a Compressor Rotor with Flow Control.	84
Figure 3.17. Dissipation Function over Temperature at 50% Span, Dry Compression.....	103
Figure 3.18. Dissipation Function over Temperature at 50% Span for 1% Wet Compression.	103
Figure 3.19. μ_{eff} at 50% Span, Dry Compression.....	107
Figure 3.20. $\frac{\Phi}{\mu_{eff}}$ at 50% Span, Dry Compression.....	107
Figure 3.21 . $2\frac{\partial u}{\partial x} - \frac{2}{3}(\bar{\nabla} \cdot \vec{u})$ at 50% Span, Dry Compression.....	108

Figure 3.22. $2 \frac{\partial v}{\partial y} - \frac{2}{3} (\bar{\nabla} \cdot \vec{u})$ at 50% Span, Dry Compression.	108
Figure 3.23. $2 \frac{\partial w}{\partial z} - \frac{2}{3} (\bar{\nabla} \cdot \vec{u})$ at 50% Span, Dry Compression.	109
Figure 3.24. Velocity Profile Sampling Location.	109
Figure 3.25. Velocity Distribution in the Boundary Layer on the Blade Suction Side at the Trailing Edge, Dry Case.	110
Figure 3.26. Velocity Distribution in the Boundary Layer on the Blade Suction Side at the Trailing Edge, Wet Case.	110
Figure 3.27. Velocity Magnitude $(\vec{W})$ Distribution for Dry and Wet Cases.	111
Figure 3.28. Magnitude of Relative Velocity at Mid-Chord of the Pressure Surface, 50% Span, Wet Case.	111
Figure 3.29. $\frac{\Phi_{AVG}}{T}$ in the Log Portion of the Casing Endwall Boundary Layer for the Dry Case, at Midspan.	114
Figure 3.30. $\frac{\Phi_{AVG}}{T}$ in the Log Portion of the Casing Endwall Boundary Layer for the 0.5% Wet Case with Tip Injection, at Midspan.	114
Figure 3.31. Distribution of y_{wall}^+ along Solid Surfaces, Dry Case (Arrows point to regions where $y_{wall}^+ \leq 11.225$).	116
Figure 3.32. Distribution of y_{wall}^+ along Solid Surfaces, Wet Case (Arrows point to regions where $y_{wall}^+ \leq 11.225$).	116
Figure 3.33. Square of the Wall Shear Stress on the Blade Suction Surface (View from Casing), Dry Compression.	120

Figure 3.34. Square of the Wall Shear Stress on the Blade Suction Surface (View from Casing), 1% Wet Compression.	120
Figure 3.35. Volumetric Rate of Entropy Production $\left[\frac{W}{K \cdot m^3} \right]$ in Zones along the Span....	121
Figure 3.36. Rate of Entropy Generation per Volume $\left[\frac{W}{K \cdot m^3} \right]$ of the Mixture due to Inter-Phase Energy Coupling.....	123
Figure 3.37. Rate of Entropy Generation per Volume $\left[\frac{W}{K \cdot m^3} \right]$ in the Rotor due to Energy Exchange between Phases.....	124
Figure 3.38. Variation of Total Temperature Ratio with Droplet Size, Uniform 1% Wet Compression Case.....	127
Figure 3.39. Relative Humidity Contours at 50% Span, $5\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.	129
Figure 3.40. Relative Humidity Contours at 50% Span, $10\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.	129
Figure 3.41. Relative Humidity Contours at 50% Span, $15\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.	130
Figure 3.42. Relative Humidity Contours at 50% Span, $20\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.	130
Figure 3.43. Evaporative Area of Sprays as a Function of Particle Diameter (Injected Mass Flow 0.005 kg/s)	133
Figure 3.44. Convective Mass Transfer Coefficient as a Function of Particle Diameter (Zero Relative Velocity Assumed).	133

Figure 3.45. Cumulative Distribution Function of Injected Spray.	137
Figure 3.46. Comparison of Total Pressure Ratio Profiles for the cases.	138
Figure 3.47. Comparison of Total Temperature Ratio Profiles.	138
Figure 3.48. Averaged Static Pressure as a Function of Axial Coordinate at Casing.....	139
Figure 3.49. Effect of Droplet Initial Temperature on the Total Temperature Ratio Profile.	141
Figure 3.50. Relative Humidity Contours at 50% Span for Water to Air Mass Flow Ratio of 1%.	143
Figure 3.51. Relative Humidity Contours at 50% Span for Water to Air Mass Flow Ratio of 1.4%.	143
Figure 3.52. Total Temperature Ratio Profiles Corresponding to Different Injected Water to Air Ratios.	144
Figure 3.53. Averaged Static Pressure as a Function of Axial Coordinate at Casing, Throttling Case.....	147
Figure 3.54. Averaged Static Pressure as a Function of Axial Coordinate at Casing.....	147
Figure 3.55. Axial Momentum as a Function of Injected Particle Diameter, Tip Injection Case.	149
Figure 3.56. Particle Traces Colored by Slip Reynolds Number, $1\mu m$ Case.....	154
Figure 3.57. Particle Traces Colored by Slip Reynolds Number, $5\mu m$ Case.	154
Figure 3.58. Particle Residence Time [s], Tip Injection with $1\mu m$ Particles.	156
Figure 3.59. Particle Residence Time, Tip Injection with $5\mu m$ particles.....	156
Figure 3.60. Computed and Measured Speedlines for NASA Rotor 37.....	159
Figure 3.61. Speedlines for Different Wet Compression Cases and Dry Case.....	159
Figure A.1. Circumferentially Averaged Total Pressure Ratio for NASA Rotor 37.....	177

Figure A.2. Circumferentially Averaged Isentropic Efficiency for NASA Rotor 37.....	177
Figure A.3. Circumferentially Averaged Total Pressure Ratio for NASA Rotor 37, Convergence Study.	179
Figure A.4. Circumferentially Averaged Total Temperature Ratio for NASA Rotor 37, Convergence Study.	179
Figure A.5. Illustration of the Pipe Test Case.....	182
Figure A.6. Control Volume Analysis of the Pipe.....	182

LIST OF TABLES

Table 3.1. NASA Compressor Rotor 37 Computed Results Summary.	78
Table 3.2. Rate of Entropy Generation $\left[\frac{W}{K} \right]$ and Calculated Isentropic Efficiency of the Rotor for Different Compression Cases.	117
Table 3.3. Rate of Entropy Generation [W/K] on Solid Surfaces.	119
Table 3.4. Inter-Phase Mass- and Momentum Sources for Different Injection Cases.	150
Table 3.5. Variation of Axial Momentum at Several Rotor Aerodynamic Locations.	152

CHAPTER 1. INTRODUCTION

1.1. Gas Turbine Evaporative Cooling Basics

Perhaps one of the most influential parameters affecting the performance of gas turbines is the temperature of the surrounding air. The net power output from a Brayton-cycle can be expressed as

$$P = \dot{m} \left[\eta_{mech} \cdot \eta_t \cdot c_p T_{t3} \left(1 - \frac{1}{\pi_{cycle}^{\frac{\gamma-1}{\gamma}}} \right) - \frac{1}{\eta_c} c_p T_{t1} \left(\pi_{cycle}^{\frac{\gamma-1}{\gamma}} - 1 \right) \right] \quad (1.1)$$

where η_{mech} is the mechanical efficiency of the shaft connecting the turbine and compressor, π_{cycle} is the pressure ratio of the cycle, η_t and η_c are the isentropic efficiencies of the turbine and the compressor, T_{t3} and T_{t1} are the turbine and compressor total inlet temperature, respectively [1]. The temperature of the ambient air impacts two parameters in Eq. (1.1): the mass flow through the machine, and the compressor inlet total temperature.

With rising compressor inlet temperature, the mass flow through the gas turbine decreases, due to the drop in air density at the compressor inlet. In addition, the compression process now starts from a higher temperature, which leads to the decrease of the output of the cycle, assuming that the cycle pressure ratio, shaft mechanical efficiency, combustion chamber inlet temperature and isentropic efficiencies of both turbine and compressor remain unchanged. Inlet air cooling is widely used to increase the output of gas turbines during summer months. This is the season when a power plants' demand for electricity is usually the greatest. Jet engine thrust decreases with rising temperatures too, making the takeoff roll distance significantly longer.

The concept of inlet air-cooling is rather simple. The air entering the gas turbine inlet is cooled before reaching the compressor inlet. This results in an increase of mass flow through the gas turbine due to the rise of the air density entering into the machine. As a result, the useful net output of the gas turbine cycle increases, which in turn translates to a thrust increase of the jet engine. If water is used for cooling, the stream increases mass flow through the engine by physically passing the air-water mixture through the engine itself, causing augmentation in net output. Additionally, with inlet cooling the compression process begins from a lower temperature, which, assuming the same cycle parameters, leads to lower compressor specific work.

From the perspective of the compressor, the increase in mass flow results in a shift of the operating point on the compressor map towards higher mass flow rates.

One of the inlet air cooling techniques is the concept of evaporative cooling, also known as inlet fogging or misting, which has been widely used in the past due to its low implementation costs. The concept is to cool down the air using evaporation of water particles injected into the compressor inlet, as described earlier. At the end of the evaporative cooling process a cooler but more humid air enters the compressor. In order to convert injected water particles into vapor, an amount of energy proportional to the latent heat of evaporation is needed, which is drawn from the air. Thus, the mixture is cooled down by phase change, meaning that the sensible enthalpy of the air decreases, while the latent enthalpy increases.

The evaporative cooling process is shown on a psychrometric chart in Figure 1.1. Dry ambient air at state A enters the gas turbine inlet, where it is sprayed with atomized water droplets. As a consequence, the dry bulb temperature of the water decreases while its humidity increases, until it finally reaches state B. The minimum theoretically achievable mixture temperature by means of evaporative cooling is the wet bulb temperature of the gas at a given

pressure, which corresponds to a fully saturated mixture. The greater the difference between dry bulb and wet bulb temperatures of the humid air, the more potential exists for evaporative cooling. This implies the lower right quadrant of the psychometric chart as favorable condition for evaporative cooling, i.e. hot and dry conditions (low relative humidity).

The evaporative cooling process can be regarded as an adiabatic saturation of the air, since the heat transfer between the stream of air entering into the compressor and the surroundings is negligible. The constant wet bulb temperature and enthalpy lines almost coincide on the psychometric chart, implying that the enthalpy as well as the wet bulb temperature of the mixture can be assumed to be constant during the process.

The cooler effectiveness, which is very useful to compare the performance of evaporative cooling equipments, is usually defined as

$$\eta_{cooler} = \frac{T_{a,db} - T_1}{T_{a,db} - T_{w,db}} \quad (1.2)$$

where T_1 is the air temperature at the compressor inlet, $T_{a,db}$ and $T_{a,wb}$ are the dry and wet bulb temperatures of the surrounding air, respectively. Typically, this number is around 80-90% for most coolers, but in some cases, it can reach up to 95%, depending mainly on atomizer characteristics. In order to achieve 100% effectiveness, full evaporation of water particles should occur before they reach the compressor.

The amount of water needed to cool down the air to a certain temperature can be calculated knowing the psychometric properties of the atmospheric air at a certain plant location. As an example, consider the following meteorological conditions at an imaginary gas turbine installation: ambient temperature of 25 degrees Celsius and a relative humidity of 50% at standard atmospheric pressure. The state point corresponding to these thermophysical properties

can either be calculated using equations of air psychometrics or read from the psychometric chart directly. In the later case, one can find the intersection of the two lines corresponding to these two properties on the chart and then read the corresponding wet bulb temperature as 17.81 °C and humidity ratio as $0.0098 \frac{kg}{kgDA}$. This means that the theoretically achievable minimum air temperature with evaporative cooling is 17.81 degrees Celsius, provided that the cooler effectiveness is 100%. The intersection of the wet bulb temperature line corresponding to 17.81 °C and relative humidity of 100% yields the properties of the humid air at the end of the evaporative cooling process. At this point, the dry bulb temperature of the mixture is equal to the wet bulb temperature, which is again 17.81 °C. The corresponding humidity ratio reads as $0.0127 \frac{kg}{kgDA}$. The amount of water needed to perform the evaporative cooling proportional to the difference between the humidity ratios at the beginning and the end of the thermodynamic process:

$$\Delta W = W_2 - W_1 = 0.0029 \quad (1.3)$$

This means that in order to cool down the air to the wet bulb temperature one needs to inject 0.0029 kilograms of water into every one kilogram of dry air.

Evaporative inter-cooling is another technique used to boost gas turbine performance. Sometimes, this procedure is called overspray or “wet compression” in the technical literature. The later term will be used here, since it describes, and at the same time characterizes, this technique of evaporative cooling. This method consists of purposefully injecting more particles into the air stream than can evaporate under the specific ambient conditions before reaching the compressor. On one hand, this provides the air density increase described previously due to

particles evaporating upstream of the compressor inlet, thereby increasing the mass flow of the humid air entering into the gas turbine. On the other hand, the reduced size droplets enter the compressor, where they continue to evaporate. This produces an inter-cooling effect on compression. Thus, the sensible heat contained in the humid air at the beginning of the wet compression is converted into latent heat content (vapor), but unlike in case of other cooling techniques, is not rejected from the thermodynamic cycle.

Along with the earlier mentioned advantages of evaporative inter-cooling, there are certain disadvantages of this procedure, associated mostly with the presence of water droplets in the compressor [2]. Depending on atomizer characteristics, place of water injection and some other factors, droplet gluing and impingement were occasionally observed on blades and casings during operation. These can cause erosion of blades, fatigue and corrosion. Additionally, in case of multi-stage machines, water injection can cause an increase in the flow coefficients for individual stages, which can potentially cause compressor surge. In case of land-based installations, this problem has only been observed in the past with excessively large amounts of water injected into the compressor, when injected water to air mass flow ratio was in excess of 3-4%. Thus, the amount of water injected into the gas turbine must have a direct impact the stationary compressor operation.

A typical land based gas turbine installation with inlet fogging is illustrated in Figure 1.2. For case of inlet fogging, nozzles are used to spray atomized water droplets directly into the inlet air stream of a compressor diffuser. The nozzle array is connected to a high-pressure pump supplying millions of droplets. The water introduced into the inlet air stream is atomized, creating a spray with large evaporative area of the droplets. The evaporative efficiency is a strong function of the quality of the spray, as mentioned previously. Thus, the finer the spray, the better is the

evaporative effect. The nozzles located at the compressor inlet spray the water into the air at high velocities, creating evaporative-inducing turbulence; inlet foggers can therefore achieve high saturation efficiency. Small droplets evaporate first cooling down the air before it enters into the compressor.

While water injection is currently used to increase the power output of land based gas turbine installations to produce electricity, it has also been applied in aircraft engines in the past, with the aim of increasing the thrust on a certain portion of the flight envelope [3]. Future applications will focus on using water injection for reducing nitrogen oxide emissions. Since engine thrust is directly proportional to the air mass flow through the engine core, evaporative cooling is an effective way to cool down the air in the engine diffuser. Typically, in these cases water is injected into the high pressure and low pressure compressor inlets. Increased thrust might mean lower engine throttle setting to achieve the same thrust, meaning less fuel consumption during take-off and climb, where fuel consumption is otherwise significant because of the dense atmosphere. In addition, with the increase of engine thrust the distance of the take-off roll is shortened, assuming same aircraft take-off weight.

Figure 1.1. Evaporative Cooling Process on Psychrometric Chart.

Figure 1.2. Gas Turbine Installation with Inlet Cooling (courtesy of Mee Industries Inc.).

1.2. Previously Published Work

1.2.1. Effects of Water Injection on Gas Turbine Operation

Recently the behavior of multi-phase flows has been drawing increasing attention in turbomachinery research. In particular, one area of interest has been the effect of ice accretion on aircraft engines, primarily due to concerns of mechanical damage and compressor stall. Ingestion of water particles by aircraft engines, in both super-cooled and liquid phases can occur in many situations during flight, for example in winter snowstorms, as well as under rainy or foggy meteorological conditions.

Chaker et al. [2] examined the effect of ambient climatic conditions on evaporation rate, and the location of the fog nozzle with respect to the gas turbine inlet duct has been studied analytically and experimentally. An analytical model was used to study the evaporation dynamics of fog droplets injected in the inlet ducts. The model was validated experimentally in a wind tunnel. The subject of sprinkle (large droplet formation) from the nozzles was also examined in detail and was shown to be non-significant. According to the authors, in some gas turbine installations for every 1°C rise in ambient temperature there was as much as 0.54% to 0.90% drop in useful power output observed.

Daggett [3] summarized operational experience gained with aircraft engines with water injection equipment. According to the author, under favorable conditions up to 20% increase of engine thrust could be achieved by the injection of only 1% water to air mass fraction into the engine core. Water injection was shown to lower the combustor inlet temperature, which resulted in higher life of the engine hot section.

While there are many advantages of water injection, there are certain disadvantages too. On several occasions, compressor icing [4], compressor blade degradation due to erosion, and blade failures due to fatigue stress [5] were observed in gas turbines operating with evaporative inlet cooling devices. The most effective way to prevent ice formation is to make sure that full evaporation of water droplets occurs upstream of the compressor inlet.

Klepper, et al. [6] and Murthy, S. N. B. [7] simulated the operation of gas turbines under wet compression conditions by coupling vaporization codes with a one-dimensional mean line compressor performance codes. According to their results, water ingestion of gas turbines has an effect on compressor performance and possibly on stability margin. Because of steam ingestion there is a change in blade incidence angles for most of the compressor stages, which results in the compressor operating point moving closer to stall.

Tsuchiya, T., et al. [8] presented a calculation procedure for the purpose of the simulation of the effect of water ingestion on axial compressor performance. In this study, the authors used the mean line stage stacking method coupled with a one dimensional vaporization code. The mean-line predicted performance for the mass flow at 80% and 90% design speed values corresponded very nearly to the experimental data obtained at that spanwise section and speed.

Meher-Homji, C.B., et al. [9-11] examined the theory of direct water fogging and discussed many practical considerations relating to the locations of the nozzles, water quality, system operation and maintenance issues. It is important for the fogging unit not to cause inlet temperature and pressure distortion, due to the susceptibility of the compressor to stall or surge in case of severe temperature unevenness. Therefore, the spatial distribution of spray has to provide relatively uniform inlet temperature and pressure distribution at compressor inlet.

Chaker, M., et al. [12], [13] provided an extensive theoretical analysis of heat and mass

transfer models for droplets, along with the analysis of the energy equation of the fluid and droplets. During the design and evaluation of inlet fogging systems, several factors need to be considered. Some of these are: methods to avoid icing at the inlet, methods to ensure intake temperature uniformity, droplet size of fogging system produced to avoid blade erosion, potential for surge, vibration of blades and gas turbine performance deterioration.

Loebig, J., et al. [14] performed a numerical study of water-methanol injection into a multi-stage axial compressor using a 3-D streamline curvature method to solve the conservation equations. The equations of motion for the injected particles were calculated by integration of droplet equations of motion in the Lagrangian frame of reference, with a two-way interaction model to simulate property exchange between phases. Authors of this paper studied the effects of spray characteristics on a 14-stage compressor performance. Results obtained by simulation were successfully validated against available experimental data. It was found that injecting particles of 10-micron diameter or less fully evaporate inside the compressor. In this case, the measured reduction in compressor static temperature at the outlet was 165 degrees F, with only 1% injected liquid to air mass flow ratio.

Ulrichs, et al. [15] studied injection of water particles into a compressor cascade using a wind tunnel. Phase Doppler Anemometry was used to measure velocity and droplet sizes during the experiment. Results show reduced air velocity around compressor airfoils due to the impact of large, high inertia droplets on the momentum of the airflow. A change in the flow angle at the outlet was also been observed. This change appeared to be function of the injected water-droplet size and mass flow.

Mund, C. F., et al. [16] presented a numerical study of a compressor washing system for a gas turbine operating under design and off-design conditions. Water injection into a compressor

inlet duct was modeled with Fluent CFD and particle trajectories were calculated. The effect of gas turbine operating conditions on particle spray patterns, injection velocities and sizes were investigated. Droplet mass and velocity were found to be the most important parameters influencing efficient blade washing, along with particle injection direction and location. Thus, particles with lower injection velocity were significantly deflected by the bulk phase, while particles with higher injection momentum were less affected by the main airflow.

Mathioudakis [17] studied the effect of water injection in the combustion chamber by means of order of magnitude analysis and thermodynamic cycle theory. It was found that water to fuel ratio is the injection parameter with the strongest influence on gas turbine performance.

Q. Zheng, et al. [18], [19] established a Moore-Greitzer stability model for wet compression using modern control theory for linear systems of equations (state-space systems stability theory). Their results show that water injection is not only a technique to improve thermodynamic performance, but also a method to stabilize the compression system when it approaches stall.

S. Jolly [20], [21] presented the thermodynamic benefits of wet compression and examined the changes in compressor and turbine operation due to water injection. Erosion is present in nearly all gas turbines operating with wet compression, because larger water droplets tend to impact compressor blades. Operational experience had shown that most erosion tends to occur near the leading edge area of the rotor blades of the first compressor stage. According to the author, one way to try to minimize erosion is to use the latest nozzle designs and reduce droplet size.

Sexton [22] presented results of a computational study that analyzes the effect of water injection on compressor performance. Compressor performance was examined while varying the

ambient temperature, humidity, water to air ratio, and water temperature. With just 0.38% water to air ratio, a 7.2 K temperature drop of the inlet air can be achieved by water misting on a 311K, 60% relative humidity day.

Wang [23] investigated the effect of water injection on the fluid mechanics of air in compressor entrance ducts. These inlet geometries included a diffuser, a bend and a straight channel, among others. For a particular diffuser authors used in the experiment, injecting particles of the diameter of the order of magnitude 30 microns or more resulted in flow separation in the diverging part, causing reverse, higher temperature airflow to enter from the back. Thus, the cooling effect of the water misting in this case decreased substantially. In the opinion of the authors, secondary flow mixing in 90-degree bend pipes had a positive effect on cooling effectiveness. Particles in this case tended to move to the wall of the pipe with higher radius, because secondary flows and centrifugal force acted on them. The authors also thought that recirculation regions must be removed, due to their tendency of trap particles, which can result in compressor erosion.

K. Hartel [24] studied the effects of evaporating droplets on compression thermodynamics, modeling the mass and heat exchange between the discrete and gaseous phases. His results showed that the most efficient way to increase the benefits of wet compression is to decrease the evaporating droplet diameter, thus increasing droplet evaporation rate.

Z. Mustafa, et al. [25] investigated trajectories of water particles in a multi-stage axial compressor using a three-dimensional RANS solver. The authors assumed one-way, aerodynamic coupling between continuous and discrete phases, focusing on droplet flow pattern in the axial compressor. Droplets were injected at the inlet of the axial compressor, assuming uniformly distributed streams along the inlet surface. Their results show that droplet flow pattern and

residence time is a strong function of the initial conditions of the discrete phase. Radial migration of droplets of larger diameter toward the casing was observed due to the centrifugal force acting on them. The droplet evaporation rate was shown to be strongly related to the initial droplet size.

A. Hamed, et al. [26] performed a numerical study of ice particle trajectories and collections on a transonic rotor at different speeds. The authors used an Eulerian-Lagrangian model to formulate governing equations of droplets and flow, using a one-way interaction model to simulate the effects of momentum and energy exchange between the different phases. Results of the flow field and particle trajectory are presented for different rotor speeds. Rotor speed was found to have a significant effect on droplet trajectories, impingement and droplet collection efficiencies. The droplet temperature at the outlet increased with the reduction in rotational speed of the rotor due to their increased residence time in the rotor.

K. Das, et al. [27], [28] conducted numerical investigations to study super-cooled water droplet trajectories and collection efficiencies on a high bypass ratio engine fan rotor under different rotational speeds. All simulations in their study used a one-way coupled Eulerian-Lagrangian multi-phase model, where particle trajectory is influenced by drag only. Their results show that the effect of rotational speed as well as droplet size had a significant effect on droplet trajectories and impingement locations. High collection efficiencies were predicted on the blade pressure surface near tip, especially in case of larger particles.

1.2.2. Efficiency of Wet Compression

A complex physical interaction of different phases, as well as interaction of droplets causes a wet compression process to be fundamentally different from a dry compression process. Consequently, defining the thermodynamic efficiency of a wet compression process is not a simple task. Published research results dealing with aspects of the entropy, energy and efficiency analysis of the wet compression in gas turbines seem to be very limited.

Zheng et al. [29], [30] discussed the thermodynamic model of wet compression process. The polytrophic exponents of the ideal and actual wet compression processes were derived using the Gibbs equation and compared to each other. With the help of T-S diagrams authors showed that wet compression work is substantially lower than the work of actual compression of air, due to the heat absorbed by the evaporating water phase. In their work, the equivalent isentropic efficiency was defined as the ratio of the ideal dry compression work to the wet compression work. Because of the decrease of the compression work in wet compression case compared to dry compression case, the equivalent adiabatic efficiency can be greater than 100%, just like in case of thermodynamic processes that involve cooling in general. The authors also defined two additional efficiencies in order to evaluate the wet compression processes.

Meacock et al. [31] defined the equivalent polytrophic efficiency for the wet compression process, which establishes the relationship between the aerodynamic increase of entropy and the polytropic efficiency of the compressor.

1.2.3. Active Compressor Stall Management

There are several techniques available to widen the operational range of compressors. These methods usually involve the introduction of a high-energy stream of air into low energy flow regions of the compressor using a second inlet. None of them considers the injection of water into the compressor as a method to extend operational range.

One such technique is boundary layer suction, where low momentum boundary layer fluid is removed from the rotor. This technique increases the critical incidence angle of the airfoil by delaying the separation of laminar and turbulent boundary layers. This is the direct result of suction making the boundary layer downstream of the suction slot thinner, which has an effect of reducing skin drag [32]. Due to decreased blockage, in this case the compressor isentropic efficiency is expected to improve too.

Other techniques are available to extend the operating range of compressors, among them tip injection. The basic idea is to minimize the blockage in this region by injecting high-momentum fluid into it. [33-38]. In all cases, air is injected from a slot that is located on the casing, usually half-one chords upstream of the tip leading edge. This way, up to 17% reduction of stall flow coefficient was observed during the experiments, along with the rise in supplied total pressure ratio.

1.3. Objectives and Scope of the Current Study

As shown, there were several papers published that examined the effect of water injection on performance of gas turbine installations. Most of these summarize past operational experience with these types of installations, while others model the interaction of different phases in multi-stage compressors by coupling one-dimensional evaporation codes with compressor mean line calculations.

To the knowledge of the author, there is no report available in the open technical literature discussing detailed flow physics and thermodynamics of transonic axial compressors with water ingestion. One of the main interests of this study is the change of the rotor flow field due to two way mass- momentum- and energy transfer among the different phases. For example, the interaction of water droplets and shock waves is very complex and the detailed flow physics has not been explored. If evaporation was enhanced due to the strong acceleration brought by the shock, it could help with the inter-cooling effect of the compression. In addition, spray characteristics affect the local flow field and the effectiveness of the cooling of the flow in the rotor.

The approach of injecting water or any other liquid (instead of air) into low-momentum areas of compressor with the aim of delaying stall has never been tried before. If phase change happens in the tip area, it should cool down the fluid in this area, resulting in higher density of flows locally, which could potentially increase momentum. In the traditional flow control techniques employed in the industry, the re-energization of momentum in low energy fluid areas has been the result of injecting high-velocity fluid into turbomachines, typically in the casing

endwall zone. In the case of water injection, however, the momentum increase could be achieved due to cooling brought by the two-way interaction of phases.

The papers dealing with the thermodynamic efficiency of the wet compression process is very limited. In all cases, authors do not calculate the efficiency using an entropy based approach of the calculation. Since entropy generation rate at a given point shows the magnitude of loss generation, it should be a particularly convenient way to assess high loss producing mechanisms present in the flow field.

The main objectives of the current study were the following:

- 1) Examine the change in rotor flow physics with water injection.
- 2) Determine the change in the aerodynamic performance parameters of the transonic rotor due to water injection at the inlet, at design conditions. Clarify the change in behavior of loss-producing mechanisms due to water injection.
- 3) Examine the effect of spray characteristics (such as initial droplet diameter, temperature of water supplied, etc) on the rotor aerodynamic performance.
- 4) Determine the speedline corresponding to cases with water injection at rotor design conditions. Examine the effect of evaporative cooling on compressor operational stability, especially for cases where water is injected into the tip region.

This includes the determination of the operating back pressure at which numerical stall will occur.

In the assumption of the current study, the shape of the water particles injected into the compressor is spherical. Furthermore, it is also assumed that particle surface is smooth and particles are non-rotating. Dilute flow is assumed, where particle motion is controlled by aerodynamic forces only (in this case drag force), as opposed to dense flows where motion of particles is also controlled by particle collisions. Two-way coupling between phases is assumed, with three modes of inter-phase property exchange present: mass, momentum and energy.

All studies were performed using NASA Transonic Compressor Rotor 37, which was designed and tested in the 1970's by Reid and Moore at NASA Lewis Research Center (currently NASA Glenn Research Center at Lewis Field). It has 36 blades, a pressure ratio of 2.106 and a mass flow of 20.19 kg/s at design conditions. The rotor speed is 1800 rad/s under design conditions, resulting in a tip speed of 454 m/s. This is a well-documented and extensively used test for RANS validations, with extensive measurement data available [39], [40]. In all cases, Fluent 6.1 was used to conduct numerical studies.

The organization of the thesis is described in the rest of this section.

In the next chapter, the governing equations of the droplets and the mixture will be introduced and interpreted. The numerical techniques used to solve these equations will also be presented, along with the two-way interaction model between the different phases.

In Chapter 3, the results of the current investigation are presented and discussed.

Section 3.1 examines the effect of water injection at the rotor inlet on its operation, at design conditions. This includes comparison of computed circumferentially averaged quantities at

the rotor outlet for wet and dry compression; also comparison of rotor operational parameters including mass flow, pressure ratio, and specific work for the two previously mentioned cases. Spanwise relative Mach number plots are also presented at several locations along with droplet trajectories to understand better the interaction of the two phases inside the rotor.

In Section 3.2 a new, entropy-based method to calculate the isentropic efficiency of the wet compression process is developed, which involves the analytical integration of wall functions in the inner parts of the endwall boundary layers and numerical integration of the entropy transport equation of the mixture in other regions of the rotor. Local flow features generating significant losses in case of wet compression are also identified.

Section 3.3 presents a parametric study of the spray parameters on the wet compression process. One injection parameter is changed at a time, while others are held constant. These include injected droplet diameter, temperature, and mass flow at the rotor inlet.

In Section 3.4, calculated speedlines for several wet compression cases are calculated and compared against the dry compression case. The mechanisms affecting the change of stable operational range of the compressor numerical stall in case of evaporating water particles present are examined in detail.

Chapter 4 discusses conclusions and future work. Several appendices are also included.

Appendix A contains the details of the numerical experiments used for the validation of the multi-phase solver that was used in this investigation.

In Appendix B, an expression of the dependence of inter-phase energy exchange on the initial particle temperature is derived. Fluent solution settings are presented in Appendix C. Appendices D and E present the source codes for grid file conversion and circumferential averaging.

CHAPTER 2. METHODOLOGY

In this chapter, the governing equations of both phases are included for completeness. The details of derivations of these can be found in Crowe et al. [41], with well-known assumptions cited in Section 1.3. These are then discretized and solved in Fluent.

2.1. Governing Equations

2.1.1. Equations of the Discrete Phase

In the Eulerian-Lagrangian multi-phase modeling approach, the droplet equations of motion are integrated with respect to time in order for the droplet trajectories to be determined. To derive the equation of motion of a particle, the acceleration of the particle is balanced by the aerodynamic drag. It can be expressed as [41]

$$\frac{d\vec{u}_p}{dt} = \frac{1}{2} C_D \frac{\pi D_p^2}{4} \rho (\vec{u} - \vec{u}_p)^2 \quad (2.1)$$

where the drag coefficient of the particle is C_D , D_p is the droplet diameter, $\vec{u}$ and $\vec{u}_p$ are the velocity vectors of the mixture and the droplet, and ρ is the density of the continuous phase. The velocity vectors are defined the following way in three-dimensional Cartesian coordinates:

$$\vec{u}_p = \begin{bmatrix} u_p \\ v_p \\ w_p \end{bmatrix}, \quad \vec{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix} \quad (2.2)$$

The particle trajectory is described by a parametric equation as a parametric curve in the three-dimensional Cartesian space:

$$\frac{d}{dt} \begin{bmatrix} x_p(t) \\ y_p(t) \\ z_p(t) \end{bmatrix} = \vec{u}_p \quad (2.3)$$

where $x_p(t)$, $y_p(t)$, $z_p(t)$ are the coordinates of the particle at any instant as a function of time t . If there is a need for the trajectories to be determined in the relative frame of reference, additional terms are applied to the equations of motion of the discrete phase. The left hand side of Eq. (2.1) in terms of relative velocities then becomes [42]

$$\frac{d\vec{u}_p}{dt} = \frac{d\vec{w}_p}{dt} + 2\vec{\Omega} \times \vec{w}_p + \vec{\Omega} \times \vec{\Omega} \times \vec{r} \quad (2.4)$$

where $\vec{\Omega}$ is the angular velocity vector of the rotating frame expressed in the inertial frame of reference, and $\vec{w}_p$ is the particle velocity in the relative frame of reference. The last two terms of Equation (2.4) account for the effect of the Coriolis and centrifugal forces acting on the particle, respectively.

According to previous investigations [14], [25], centrifugal force does not affect significantly the trajectory of liquid droplets in axial compressors, unless the particle diameter is greater than 50-100 microns.

The drag coefficient C_D on the right hand side of Eq. (2.1) is usually determined from

empirical correlations involving the droplet slip Reynolds number based on the relative velocity between droplet and gas. This is defined as

$$\text{Re}_D = \frac{\rho D_p \|\vec{u} - \vec{u}_p\|}{\mu} \quad (2.5)$$

where μ is the dynamic viscosity of the continuous phase.

Figure 2.1 shows the variation of the measured drag coefficient of a sphere as a function of the Reynolds number based on particle slip velocity. For the limit of low slip Reynolds numbers, the inertial terms of the Navier-Stokes equations describing the flow around the sphere are negligible. Under these conditions, the drag coefficient can be approximated by the well-known Stokes law [41]:

$$C_D = \frac{24}{\text{Re}_D} \quad (2.6)$$

The variation of the drag coefficient with the slip Reynolds is approximately linear in this flow regime, intersecting the y-axis at the same point as the measured curve. The drag coefficient drops with increasing slip Reynolds numbers, becoming almost constant from Re_D of approximately 800 until about 30000. This operating region is called inertial range, because inertial forces acting on the droplet become significant, while the viscous force unimportant. As the curve flattens out at around Reynolds number of 800, the particle drag is almost entirely due to form drag, which is the result of the asymmetry of the pressure forces acting on the particle, with shear drag contributing little. The compressibility effects on the momentum transfer become substantial when the relative Mach number of the flow around the sphere reaches about 0.6. This is the point when the first sonic flow occurs around the sphere.

Usually, for ranges of Reynolds numbers that are greater than order of one, the coefficient of drag is expressed in the form of

$$C_D = a_1 + \frac{a_2}{Re_D} + \frac{a_3}{Re_D^2} \quad (2.7)$$

where a_1 , a_2 , a_3 are coefficients used for polynomial fitting of measurement data.

Figure 2.1. Variation of Sphere Drag Coefficient with Slip Reynolds Number.

2.1.1.1. Response Time

There are several similarity numbers characterizing the multi-phase flow fields. One such parameter is the response time of the droplet to changes in fluid velocity and temperature.

Assuming low slip Reynolds numbers (Stokes flow), the equation of motion of the particle can be written as

$$\frac{d\vec{u}_p}{dt} = \frac{1}{\tau_v} \rho [\vec{u} - \vec{u}_p] \quad (2.8)$$

where the quantity τ_v is called the momentum response time. If the particle is at rest initially, the solution of the equation of motion is

$$\vec{u}_p = \vec{u} \left[1 - e^{-\frac{t}{\tau_v}} \right] \quad (2.9)$$

meaning that any particle released into flow accelerates until reaching the speed of the carrier fluid. The momentum response time, by definition, is equal to [41]

$$\tau_v = \frac{\rho_p D_p^2}{18\mu} \quad (2.10)$$

where ρ_p is the density of the discrete phase. This is the time required for the particle to achieve 63% of the free stream velocity from rest in Stokes flow.

Similar to the momentum response time, the thermal response time relates the responsiveness of the particle temperature to changes in fluid temperature, assuming no mass exchange between the phases (i.e. no evaporation or condensation). It is defined as [41]

$$\tau_T = \frac{\rho_p c_d D_p^2}{12k} \quad (2.11)$$

where c_d is the specific heat of the discrete phase (particle material) and k is the thermal conductivity of the gas surrounding the droplet. One can think of this parameter as the time required for a particle to achieve 63% of the free stream temperature under Stokes flow

conditions. Essentially, both thermal and momentum response times are proportional to the time needed for the dispersed phase to achieve equilibrium with the continuous phase.

Figure 2.2. Variation of Momentum and Thermal Response Time with Droplet Diameter.

The particle is in velocity and thermal equilibrium condition with the fluid, if the following criteria are satisfied:

$$\begin{aligned}\vec{u} &= \vec{u}_p \\ T &= T_p\end{aligned}\tag{2.12}$$

Figure 2.2 shows the variation of momentum and thermal response times of a water particle with droplet diameter. Since the diameter is squared in both expressions, these two quantities are most

sensitive to changes of diameter. For example, the momentum and thermal response times for a droplet diameter $10\mu m$ are hundred times the response times of a $1\mu m$ diameter particle. Since larger particle takes much longer to react to changes in flow velocity, (according to Fig. 2.2), one can assume that with increasing diameter the average particle velocity across a given system decreases. This results in an increase in the residence time of a $10\mu m$ particle in a system compared to a $1\mu m$ particle. The same is true for the thermal behavior of particles: larger particle takes longer time to achieve the fluid temperature, thus, theoretically, the potential for phase change, the temperature gradient at the fluid-droplet interface will be significant for longer time. For a water droplet size of order of a micron in air, the momentum response time is of the order of 10^{-4} seconds.

2.1.1.2. Stokes Number

The other important parameter describing the droplet-fluid interaction is the Stokes number [41], which is a non-dimensional parameter describing the character of the multi-phase flow through the momentum response time. The Stokes number is defined as the ratio of the momentum response time to some characteristic time of the system,

$$St = \frac{\tau_v}{\tau_{sys}} = \frac{\tau_v U}{L} \quad (2.13)$$

where U is some characteristic velocity in the system (for example the velocity of the flow in a pipe), while L is a characteristic dimension of the system (length of the pipe). If the Stokes number is less than one, response time of the particles is much less than the characteristic time of

the flow field. In this case, particles will quickly attain the velocity of the fluid and follow the fluid closely resulting in equilibrium condition. If the Stokes number is more than one, than the particle will have no time to respond to a change of the fluid velocity within system boundaries. Consequently, the particle velocity will not be affected substantially during its residence in the flow field.

The Stokes number characterizes the dispersion of particles due to fluctuating fluid forces encountered in turbulent structures. In order to estimate the effect of turbulent dispersion on particle motion, Crowe [43] proposed to evaluate the Stokes number of the particle, which is the ratio of the particle response time to some time scale related to some time characteristic of the flow field,

$$St = \frac{\tau_v \Delta u}{l} \quad (2.14)$$

where l is the size of the turbulent structure and Δu is the velocity difference of the fluid across the shear layer.

Accurate estimation of the local flow time scale is critical for the estimation of the effect of the turbulent dispersion on particle motion, since the Stokes number is proportional to $\frac{l}{\Delta u}$. If the $k - \varepsilon$ turbulence model were used during flow solution, one way to estimate the turbulent eddy time scale would be

$$\tau_{urb} \cong \frac{l}{\|\vec{c}'\|} \cong \frac{c_\mu \frac{1}{\rho} \frac{\tilde{k}^{\frac{3}{2}}}{\varepsilon}}{\sqrt{\frac{2\tilde{k}}{3}}} \cong \frac{c_\mu \cdot \tilde{k}}{\rho \varepsilon} \sqrt{\frac{3}{2}} \quad (2.15)$$

where $\tilde{k}$ is the turbulent kinetic energy, ε is the turbulent kinetic energy dissipation rate and c_μ is a model constant. Then, one could possibly evaluate the local Stokes number according to:

$$St \cong \frac{\tau_v}{\tau_{turb}} \quad (2.16)$$

According to Zheng et al. [29], particles smaller than $5\mu m$ in diameter can follow the flow closely in gas turbines. This is because in their opinion, small particles have small response time, thus achieving velocity equilibrium with the continuous phase in a small amount of time. Loebig et al. [14] in their experiment found that droplets smaller than $25-50\mu m$ in diameter have the ability to follow the fluid very closely. Consequently, according to the findings of these authors, when modeling particles smaller the $50\mu m$ in diameter, the effect of turbulent dispersion on particle motion should be negligible.

The larger the particle, the larger is its inertia, and therefore the inability to follow the turbulent eddies closely, which – for general cases - can be quantified by the evaluation of the particle Stokes number.

2.1.1.3. Mass and Energy Transfer through Evaporation

In addition to the momentum coupling due to drag, the particle also interacts with the continuous phase by means of mass and energy coupling. The mass and energy transfer between the two phases occurs at the droplet-mixture interface. Due to difference in temperature and water vapor concentration between the free stream and droplet-air interface, the particle is surrounded by thermal and diffusion boundary layers. The driving potential for the mass exchange between

the discrete and continuous phases is proportional to the gradient of mass fraction of the water vapor at the interface of the droplet. During evaporation, there is a vapor mass flux through the droplet control surface. Fick's law establishes the relationship between the mass flux vector and the gradient of the vapor mass fraction of the evaporating species at the fluid-droplet interface. It can be expressed as [41]

$$\vec{J}_V = -\rho_s D_V \frac{\partial \omega_V}{\partial \vec{n}} \Big|_{r=R_D} \quad (2.17)$$

where $\vec{J}_V$ is the vapor mass flux vector, ρ_V is the density of the vapor, ω_V is the vapor mass fraction, $\vec{n}$ is the droplet control surface normal vector, D_V is the binary mass diffusion coefficient of the water vapor in air, and R_D is the radius of the particle. Essentially, for evaporation to happen, the directional derivative at the particle surface, $\frac{\partial \omega_V}{\partial \vec{n}} \Big|_{r=R_D}$ should be positive, which translates to the criteria

$$\omega_V \Big|_{r=R_D} > \omega_{V,\infty} \quad (2.18)$$

where $\omega_{V,\infty}$ is the vapor mass fraction in the free stream.

At the droplet surface, ω_V is assumed to be proportional to the partial pressure of the evaporating species, according to the universal gas law [41]

$$\omega_V \Big|_{r=R_D} = \frac{M}{\rho} \frac{p_V}{R_V T_D} \quad (2.19)$$

where p_V is the partial pressure of the vapor at the droplet surface, R_V is the gas constant of the vapor, T_D is the droplet temperature, M is the universal gas constant.

At moderate droplet slip Reynolds numbers, it seems reasonable to assume conditions at the particle surface described by the Clausius-Clapeyron relation [14], [44], and [45]. Thus, the partial pressure of the water vapor at the droplet surface is related to the saturation vapor pressure of water in air:

$$p_V = p_{sat}(T_P) \quad (2.20)$$

Furthermore, this assumption is also valid for the cases of moderate non-equilibrium for the cases where relative Mach numbers of the droplets with respect to gas are sufficiently small to neglect compressibility effects [45]. The saturation vapor pressure corresponding to a given droplet temperature and atmospheric pressure can be looked up in steam tables or calculated using curve fits of steam properties.

The mass fraction of the water vapor far away from the droplet can be determined by solving the species transport equation for water vapor with appropriate boundary conditions. This is a partial differential equation very similar to the simplified energy equation, containing a convective and a diffusive term.

The rate of change of droplet mass can be expressed in terms of the mass fraction difference between the particle and the free stream, analogously to the Fourier-law of heat transfer using the following equation [41]

$$\frac{dm_p}{dt} = Sh_D \cdot \pi \cdot D_p \cdot \rho \cdot D_V \left(\omega_{V,\infty} - \omega_{V,r=R_D} \right) \quad (2.21)$$

where Sh_D is the Sherwood number, which is the non-dimensional mass-transfer coefficient. By definition, this similarity parameter is [41]

$$Sh_D = \frac{h_c D_p}{D_V} \quad (2.22)$$

where the h_c is the mass transfer coefficient. There is similarity with Nusselt number, since the two processes are analogous. The Nusselt number is defined as

$$Nu_D = \frac{hD_p}{k} \quad (2.23)$$

where k is the thermal conductivity of the mixture evaluated at film conditions and h is the convective heat transfer coefficient.

One can estimate the lifetime of a droplet (i.e. time to complete evaporation), by rewriting Equation (2.21) [44]:

$$D_p \frac{dD_p}{dt} = -\frac{Sh_D \cdot \rho \cdot D_v}{\rho_p} (\omega_{V,r=R_D} - \omega_{V,\infty}) \quad (2.24)$$

Solving this ordinary differential equation with the boundary condition

$$D_p(t=0) = D_0 \quad (2.25)$$

gives

$$D_p^2 - D_0^2 = -\lambda t \quad (2.26)$$

where λ is called evaporation constant, (for certain applications this approximation is valid) and it is equal to:

$$\lambda = \frac{4 \cdot Sh_D D_v \rho}{\rho_p} (\omega_{V,\infty} - \omega_{V,r=R_D}) \quad (2.27)$$

If the particle diameter is assumed zero at the time when the particle evaporates, the lifetime of the particle becomes:

$$\tau_m = \frac{D_0^2}{\lambda} \quad (2.28)$$

Thus, the droplet lifetime is proportional to the square of the initial particle diameter. The Biot number is defined as [41]

$$Bi_D = \frac{hD_p}{k_p} \quad (2.29)$$

Where k_p is the thermal conductivity of the particle. If the Biot number of the particle is small, than the internal thermal resistance of the droplet is negligible. In this case, the droplet energy equation expresses the change of internal energy of the droplet, and can be written as [41]:

$$\rho_p c_p \frac{dT_p}{dt} = hA_p (T - T_p) + \frac{dm_p}{dt} h_{fg} \quad (2.30)$$

where h_{fg} is the latent heat of evaporation, A_p is the surface area, and h is the convective mass transfer coefficient at the droplet-fluid interface. The left hand side of Equation (2.30) represents the change of the internal energy of the particle, while the right hand side contains the term expressing the sensible energy exchange with the mixture through convection and the energy required for the phase change of the droplet, respectively. Since the later term also contains the rate of change of droplet mass, the inter-phase momentum and energy transfer processes are coupled.

If there is no phase change happening, the second term on the right hand side of Eq. (2.30) will vanish. In this case, the temperature of the particle will only be governed by the heat convected to or from the particle.

If the particle evaporates at wet bulb temperature, the change related to the particle internal energy will be zero. In this case, the convective heat transfer between particle and surroundings is only enough to supply the energy required for the phase change.

Finally, if there is no convective heat transfer between the particle and its surroundings, the energy required for the phase change is drawn from the internal energy of the droplet.

One can rewrite the droplet mass and energy equation in a more convenient non-dimensional form as [46]

$$\frac{dm_p}{dt} = Sh_D \cdot \pi \cdot D_p \cdot \rho \cdot D_V \left(\omega_{V,\infty} - \omega_{V,r=R_D} \right) \quad (2.31)$$

$$\frac{dT_p}{dt} = \frac{Nu_D}{2} \frac{1}{\tau_T} (T - T_p) + \frac{Sh_D}{2} \frac{1}{\tau_T} \frac{Pr}{Sc} \frac{h_{fg}}{c_p} \left(\omega_{V,\infty} - \omega_{V,r=R_D} \right) \quad (2.32)$$

where Pr is the Prandtl number, and Sc is the Schmidt number. The Schmidt number is analogous to the Prandtl number and is defined as the ratio of momentum diffusivity to mass diffusivity, calculated at film conditions:

$$Sc = \frac{\nu}{D_v} \quad (2.33)$$

The non-dimensional mass and heat transfer numbers in Equation (2.31) and Equation (2.32) can be calculated according to the Ranz-Marshall correlations [47]:

$$Sh_D = 2 + 0.6 Re_D^{\frac{1}{2}} Sc^{\frac{1}{3}} \quad (2.34)$$

$$Nu_D = 2 + 0.6 Re_D^{\frac{1}{2}} Pr^{\frac{1}{3}} \quad (2.35)$$

Equation (2.34) is valid for slip Reynolds numbers up to 200, while Equation (2.35) for slip Reynolds numbers up to 50000.

2.1.2. Equations of the Mixture

Figure 2.3. illustrates a control volume surrounding a region of a binary mixture. The control surfaces marked with dotted lines surround the particles. Thus, when particles evaporate,

the control surfaces surrounding the droplets adjust accordingly to be just outside of the droplet-gas interface. It is assumed that the number of particles residing in the control volume is equal to N . Finally, it is also assumed that a control volume V is surrounded by the surface S , and is small enough to express the conservation laws of the mixture in differential form.

Assuming that the volume fraction of the dispersed phase is much less than that of the continuous phase, the continuity equation for the mixture becomes

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{u}) = - \frac{\sum_{j=1}^N \dot{m}_{p,j}}{V} \quad (2.36)$$

where ρ is the mixture density, $\vec{u}$ is the mixture velocity vector, $\dot{m}_{p,j}$ is the mass lost by particle denoted by index j . The term on the right hand side in Equation (2.36), which is often denoted by S_{mass} in the literature, acts like a source term if a change of phase is happening. In this case, this is the mass flow added to the continuous phase by the evaporating particles in the form of water vapor. The summation on the right hand side of Equation (2.36) should be carried out for all particles surrounded by control surface S . Essentially, the sign of this source term is positive when evaporation in the control volume occurs. This is because the sign of $\sum_{i=1}^N \frac{dm_{p,i}}{dt}$ on the right hand side will be negative, since the mass of evaporating particles decreases with evaporation.

The momentum equation for the mixture surrounded by control surface S can be written in a similar manner. Assuming that the volume fraction of droplets is small compared to that of the continuous phase, the result is [48]

$$\begin{aligned}
\frac{\partial(\rho \bar{\mathbf{u}})}{\partial t} + (\bar{\mathbf{u}} \cdot \bar{\nabla})(\rho \bar{\mathbf{u}}) = & -\bar{\nabla} p + \bar{\bar{\boldsymbol{\tau}}} \cdot \bar{\nabla} + \rho \cdot \bar{\mathbf{g}} \\
& - \sum_j \rho_{p,j} \frac{f_j}{\tau_{V,j}} (\bar{\mathbf{u}} - \bar{\mathbf{u}}_{p,j}) - \frac{1}{V} \sum_j \dot{m}_j \bar{\mathbf{u}}_{p,j}
\end{aligned} \tag{2.37}$$

Figure 2.3. Control Volume Enclosing Part of the Mixture.

where $\bar{\bar{\boldsymbol{\tau}}}$ is the viscous stress tensor, f_j is the drag factor of particle denoted by index j . In three-dimensional Cartesian coordinates, the viscous stress tensor can be expressed as:

$$\begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix} \tag{2.38}$$

The drag factor, by definition, is the ratio of the sphere drag coefficient to Stokes drag:

$$f_j = \frac{C_{D,j} \text{Re}_{D,j}}{24\tau_{V,j}} \tag{2.39}$$

where it should be recalled that $\tau_{v,j}$ is the momentum response time of particle denoted by index j . There are two terms on the right hand side of Eq. (2.37), which provide the coupling between continuous and discrete phases. The last term accounts for the momentum addition caused by the vapor mass flux from the particles to the mixture. The fourth term represents the momentum source due to aerodynamic drag force, which is proportional to particle slip velocity and particle diameter. The summation in these terms should be carried out over all particles residing in the control volume bounded by the surface $\mathcal{S}$, up to N particles, since all of them contribute to the inter-phase momentum exchange.

In case of low volume fraction of dispersed phase (less than 10%), the energy equation for the continuous phase enclosed by the control volume V can be expressed as [48]

$$\begin{aligned} \frac{\partial(\rho E_t)}{\partial t} + \bar{\nabla} \cdot (\rho \bar{u} h) = & -\frac{1}{V} \sum_j \dot{m}_j h_{s,j} - \sum_j \rho_{p,j} \frac{f_j}{\tau_{v,j}} (\bar{u}_{p,j} - \bar{u}) \cdot \bar{u}_{p,j} \\ & + \bar{\nabla} \cdot (\bar{\tau} \cdot \bar{v}) + \bar{\nabla} \cdot (k \bar{\nabla} T) - \frac{1}{V} \sum_j \dot{q}_j \end{aligned} \quad (2.40)$$

where h is the specific enthalpy of the mixture, $h_{s,j}$ is the enthalpy of the mixture at the fluid-gas interface of droplet j , k is the thermal conductivity of the mixture, $\dot{q}_j$ is the rate of sensible heat exchanged between droplet j and the mixture by means of convection. The total energy is defined the following way:

$$E_t = h + \frac{|\bar{u} \cdot \bar{u}|}{2} - \frac{p}{\rho} \quad (2.41)$$

The two-way energy coupling between the phases is represented in Eq. (2.40) through all terms that have a summation sign in front of them, meaning that these terms have to be considered for all particles residing in the control volume containing the mixture. The first term on the right hand side of Eq. (2.40) accounts for the enthalpy influx into the control volume due to the

evaporation of particles, since it contains the enthalpy of the water vapor in addition to the liquid water. Essentially, this term implicitly includes the latent heat of evaporation; it will be dominant during the wet compression process, and it will increase the latent enthalpy (i.e., vapor content) of the gas. The second term on the right hand side of Eq. (2.40) is the energy loss of the gas phase due to work performed on the dispersed phase; it can be assumed to be negligible in case particles are in velocity equilibrium with the flow or even when relative slip Mach number of particles is low. The last term on the right hand side of Eq. (2.40) represents the convective heat transfer from or to the dispersed phase, which arises due to temperature difference between the droplet and the flow; it accounts for the sensible heat exchange between the droplet and the mixture. The role of this term in the evaporative cooling process is crucial, since the energy needed for evaporation is supplied by convection. All the other terms are also present in the energy equation describing single-phase flows.

Integrating the mass, momentum and energy equations of the mixture with respect to the volume of a computational cell of interest and applying Green's theorem to the appropriate terms yields to the set of partial differential equations in vector form, expressed as

$$\frac{\partial}{\partial t} \int_V \vec{W} dV + \oint_A (\vec{F} - \vec{G}) \cdot d\vec{A} = \int_V \vec{H} dV \quad (2.42)$$

Where the convective flux vectors are defined the following way in this system:

$$\vec{W} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E_t \end{bmatrix} \quad \vec{F} = \begin{bmatrix} \rho \vec{c} \\ \rho \vec{u} u + p \vec{i} \\ \rho \vec{u} v + p \vec{j} \\ \rho \vec{u} w + p \vec{k} \\ \rho \vec{u} E_t + p \vec{u} \end{bmatrix} \quad \vec{G} = \begin{bmatrix} 0 \\ \tau_{xi} \\ \tau_{yi} \\ \tau_{zi} \\ \tau_{ij} v_j + \vec{q} \end{bmatrix} \quad \vec{H} = \begin{bmatrix} S_{mass} \\ S_{mom,x} \\ S_{mom,y} \\ S_{mom,z} \\ S_{ener} \end{bmatrix} \quad (2.43)$$

The conservation equations were derived assuming the validity of the Stokes hypothesis, where the components of the viscous stress tensor are defined the following way:

$$\tau_{ij} = 2\mu \left(\frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{1}{3} (\bar{\nabla} \cdot \bar{u}) \delta_{ij} \right) \quad (2.44)$$

Here, δ_{ij} is the Dirac-delta function. The velocity vector is defined in Cartesian coordinates:

$$\vec{c} = u \cdot \vec{i} + v \cdot \vec{j} + w \cdot \vec{k} = \sum_{i=1}^3 u_i \vec{e}_i \quad (2.45)$$

Sutherland's law describes the relationship between the molecular viscosity and temperature:

$$\mu = \mu_{ref} \left(\frac{T_{ref} + T_s}{T + T_s} \right) \left(\frac{T}{T_{ref}} \right)^{\frac{3}{2}} \quad (2.46)$$

Where the reference values are equal:

$$T_{ref} = 273K$$

$$\mu_{ref} = 1.76 \cdot 10^{-5} \frac{kg}{m \cdot s} \quad (2.47)$$

$$T_s = 110.333K$$

In Eq. (2.43), the heat flux vector due to conduction is defined as:

$$\vec{q} = -k \bar{\nabla} T \quad (2.48)$$

The vector $\vec{H}$ on the right hand side of Equation (2.42) contains the mass source S_{mass} , momentum source vector $\vec{S}_{mom}$ and energy source S_{ener} terms present in the control volume due to phase change of the droplets, which is of primary interest in this investigation. These are the inter-phase mass-, momentum and energy terms present in the conservation equations derived previously.

Along with Eq. (2.42), the species transport equation is a simple convection-diffusion equation describing the local mass fraction of each species of a multi-component mixture on the domain of interest:

$$\frac{\partial}{\partial t}(\rho Y_j) + \vec{\nabla} \cdot (\rho \vec{c} Y_j) = -\vec{\nabla} \cdot \vec{J}_j + R_j + S_j \quad (2.49)$$

Here, the quantity Y_j is the mass fraction of species j at a given point of the domain of interest, $\vec{J}_j$ is the mass flux vector of species j at a given point of space. The quantities R_j and S_j are the mass rates of creation or depletion of species j due to chemical reaction and by addition from the dispersed phase, respectively. Equation (2.49) has to be solved with appropriate boundary and initial conditions to determine the mass fraction of species j on the domain of interest.

In order to close the system of governing equations, it is necessary to establish relationships between the thermodynamic variables. The gas contained by the control volume can assumed to behave as a mixture of two ideal gases, air and water vapor. Furthermore, it should obey Dalton's law of partial pressures. In order to accomplish this, the well-known equation of equation of state is used:

$$\rho = \frac{p}{\sum_{i=1}^N \frac{m_i}{M_i} RT} \quad (2.50)$$

In Eq. (2.50) R is the universal gas constant, M_i is the molecular weight and m_i is the mass fraction of species denoted by index i .

2.1.3. Turbulence Model

After applying the Favre decomposition of flow variables to Eq. (2.42), the system of governing equations can be expressed in terms of density-averaged quantities. Furthermore, the viscous stress tensor and the heat transfer term due to conduction will both have two components: one accounting to the momentum and heat transport due to mean gradients, while the other accounting for the property transport due to gradients arising from turbulent fluctuations. For example, the Reynolds stress tensor can be expressed as

$$\tau_{ij} = 2\mu \left(\frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{1}{3} \left(\frac{\partial \bar{u}_k}{\partial x_k} \right) \delta_{ij} \right) - \overline{\rho u'_i u'_j} \quad (2.51)$$

where the last term of the right hand side is the apparent stress tensor, accounting for the velocity fluctuations of the turbulent flow. The apparent stress tensor shows up in the governing equations as an unknown. In order to solve Eq. (2.42) for a given set of boundary and initial conditions, it is desirable to express the apparent stresses in terms of the mean velocity gradients. This is called the closure problem. Several one and two equation models exist in the technical literature to accomplish this task. In the current study a two-equation model, the k- ϵ model is used, which is based on the Boussinesq assumption, i.e., that the apparent stresses in turbulent flows are related to the mean flow strains the following way:

$$-\overline{\rho u'_i u'_j} = \frac{2}{3} \delta_{ij} \rho \tilde{k} - 2\mu_t \left(\frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{1}{3} \left(\frac{\partial \bar{u}_k}{\partial x_k} \right) \delta_{ij} \right) \quad (2.52)$$

The turbulence kinetic energy $\tilde{k}$ is defined as:

$$\tilde{k} = \frac{1}{2} \overline{u'_i u'_i} \quad (2.53)$$

In order to determine the turbulent viscosity the following set of transport equations are solved for turbulent kinetic energy k and dissipation rate ε :

$$\frac{\partial(\rho\tilde{k})}{\partial t} + \frac{\partial(\rho u_i \tilde{k})}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{Pr_t} \frac{\partial \tilde{k}}{\partial x_j} \right) + P_t - \rho\varepsilon \quad (2.54)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{Pr_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{C_{\varepsilon 1} P_t + C_{\varepsilon 2} \rho\varepsilon + E}{\tau_t} \quad (2.55)$$

The first term on the right hand side of Eq. (2.54) represents the transport of the turbulent kinetic energy by diffusion. The second term on the right hand side of Equation (2.54) represents the rate of turbulence production, and is equal to:

$$P_t = -\overline{\rho u_i' u_j'} \frac{\partial \bar{u}_i}{\partial x_j} \quad (2.56)$$

As one can see from this expression, the generation of turbulence energy is a result of the mean velocity gradients present in the flow. The third term represents the dissipation of the turbulent kinetic energy. The quantities Pr_t and Pr_ε are the turbulent Prandtl numbers for the turbulent kinetic energy and dissipation respectively. τ_t is the realizable estimate of turbulence time scale. The model constants have the following default values in the solver used in the current investigation:

$$C_{\varepsilon 1} = 1.44 \quad C_{\varepsilon 2} = 1.92 \quad C_\mu = 0.09 \quad Pr_k = 1.0 \quad Pr_\varepsilon = 1.3 \quad (2.57)$$

The eddy viscosity is calculated if the turbulent kinetic energy and dissipation are known:

$$\mu_t = \rho \cdot C_\mu \cdot \frac{\tilde{k}^2}{\varepsilon} \quad (2.58)$$

Once the turbulent viscosity is known at a given point and time, the effective viscosity and thus the Reynolds stress tensor can be calculated using the mean velocity gradients as

$$\tau_{ij} = 2 \cdot (\mu + \mu_t) \cdot \left(\frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{1}{3} \left(\frac{\partial \bar{u}_k}{\partial x_k} \right) \delta_{ij} \right) \quad (2.59)$$

where the sum of turbulent and laminar viscosities is called effective viscosity.

2.2. Computational Model

2.2.1. Discretization of Governing Equations

The RANS solver used in the current investigation uses a control volume approach to convert the system of conservation equations into a set of algebraic equations. This means that volume integrals are converted into surface integrals and later approximated algebraically. In order to explain the discretization process, Figure 2.4 shows the physical domain subdivided into computational polygons. The centroid of each triangular element is denoted by $\mathbf{C}$.

After the tessellation is complete, the second term on the left hand side of Eq. (2.42) can be expressed in terms of algebraic expressions the following way:

$$\oint_{(A)} (\vec{F} - \vec{G}) \cdot \vec{n} dA \cong \sum_{i=1}^N (\vec{F}_i - \vec{G}_i) \cdot \vec{n}_i A_i \quad (2.60)$$

In this expression, N is the number of faces in a computational cell, A_i is the area of the computational face, $\vec{n}_i$ is the surface normal vector of a given side of the computational face

denoted by index i . $\vec{F}$ and $\vec{G}$ are the convective and diffusive flux vectors through a given computational face. Property values are stored at cell centers during calculations. However, in order to solve the set of governing equations, face values are required for the terms containing convective and diffusive flux vectors.

Figure 2.4. Illustration of Computational Cells on the Domain of Interest.

The quantities at cell faces are computed using a Taylor Series expansion about the cell centroid:

$$\phi_{face} = \phi(\vec{r}_0 + \delta\vec{r}) = \phi(\vec{r}_0) + \vec{\nabla}\phi \cdot \delta\vec{r} \quad (2.61)$$

In Equation (2.61), ϕ is a scalar quantity, $\vec{r}_0$ is the position vector of the centroid of the computational cell, and $\delta\vec{r}$ is the displacement vector from the cell centroid to the face centroid.

The gradient in this expression is calculated by the divergence theorem:

$$\vec{\nabla}\phi = \frac{1}{V} \sum_{i=1}^N \tilde{\phi} \cdot \vec{n}_i A_i \quad (2.62)$$

In Equation (2.62), V is the volume of the computational cell, while the quantity $\tilde{\phi}$ is computed by averaging ϕ from the two cells adjacent to a given face if second order accuracy of approximation is desired.

2.2.2. Wall Function Approach

The quality of the solution of the turbulence model is strongly dependent on the grid spacing next to solid walls. If the integration of the system of governing equations is performed all the way to the wall, sometimes excessively high number of grid points are necessary in order to resolve the inner part of the boundary layer properly, since in this area the velocity gradient is usually very steep.

Usually, the distance from a wall in a turbulent boundary layer is characterized in terms of the non-dimensional characteristic coordinate defined as

$$y^+ = \frac{y \sqrt{\frac{\tau_w}{\rho}}}{\nu} \quad (2.63)$$

where $u^* = \sqrt{\frac{\tau_w}{\rho}}$ is the wall friction velocity, by definition.

Experience has shown in these cases the dimensionless distance from the wall to the adjacent cell needs to be approximately of order one to achieve a quality solution for turbulent flows. In many applications, this condition leads to excessively high number of grid points necessary in order to resolve the inner part of the boundary layer properly which can be

computationally very expensive. This is especially true for high-Reynolds number flows, where the buffer layer and the laminar sub-layer tend to be very thin.

In order to save computational time and still arrive at a reasonable solution in many cases, wall functions can be used instead of integrating the conservation equations all the way to the wall. Wall functions produce reasonable solutions even at y^+ values much larger than unity, at the expense of additional assumptions for the inner layer.

In the wall function approach, the inner part of the boundary layer does not get resolved. Instead, a wall-parallel mean velocity variation is assumed locally, with the knowledge of the flow properties in the cell adjacent to the wall, from which a wall shear stress, production of turbulent kinetic energy and dissipation rate are estimated based on certain assumptions. The former is used as a boundary condition at the walls while solving the momentum equations, while the later is used to solve the transport equations of the turbulence kinetic energy. Usually, wall functions assume that the flow very near to the wall behaves according to:

$$u^+ = \frac{u}{u_*} = f(y^+) \quad (2.64)$$

More specifically, the following wall functions are employed in the RANS solver used in the current investigation:

$$\begin{aligned} u^+ &= y^+ & y^+ &\leq 11.225 \\ u^+ &= 2.5 \cdot \ln y^+ + 5.5 & 11.225 < y^+ \leq y_{wall}^+ \end{aligned} \quad (2.65)$$

Once the velocity in a wall-adjacent cell is known, the shear stress at the wall can be calculated using the law of wall. The next step is to calculate the turbulent kinetic energy according to the k-

equation. Then, the production of turbulent energy and dissipation in the cell is calculated according to:

$$P_t = \frac{\tau_w^2}{0.41 \cdot C_\mu^{\frac{1}{4}} \cdot k^{\frac{1}{2}} \cdot y} \quad (2.66)$$

$$\varepsilon = \frac{C_\mu^{\frac{3}{4}} \cdot k^{\frac{3}{2}}}{0.41y} \quad (2.67)$$

The wall functions are valid if the grid is constructed in such a way that for the near-wall cells the criterion $0.1 \leq y^+ \leq 200$ holds [49]. For best results, for the solver used in the current investigation the recommended value is:

$$11.2 \leq y^+ \leq 60 \quad (2.68)$$

Wall functions were derived using certain assumptions regarding the state of the flow in the laminar sublayer and the inner part of the log-layers, as it was mentioned earlier. Thus, the standard wall function approach tends to provide inaccurate results when near-wall flows are subjected to large pressure gradients or in cases when flows are in non-equilibrium condition.

For the case of multi-phase flows, the k-ε turbulence models are still shown to be valid with some modifications depending on the character of the dispersed phase flow. Particles can either suppress or enhance the turbulence of the mixture depending on some characteristics of the turbulence of the carrier phase as well of that of the discrete phase. The effect of particles on the turbulence is shown to be weak if the particle concentration is low (which is true for the current cases.)

2.2.3. Boundary and Initial Conditions

Constant total pressure and constant total temperature with standard atmospheric conditions were imposed as boundary conditions at the inlet, with turbulence intensity of five percent and turbulence length scale of eight percent of the blade height. Since the conditions at the inlet are subsonic, the inflow velocity is directly extrapolated from the interior. After that, the static pressure and temperature in the inlet plane is computed using isentropic flow relations:

$$\frac{T_{t,1}}{T_1} = 1 + \frac{\gamma - 1}{2} M^2 \quad (2.69)$$

And

$$\frac{p_{t,1}}{p_1} = \left[1 + \frac{\gamma - 1}{2} M^2 \right]^{\frac{\gamma}{\gamma - 1}} \quad (2.70)$$

The static pressure at the outlet hub was set to different values in order to simulate each operating point. All other properties of the flow were extrapolated from the interior domain. Additionally, the radial equilibrium boundary condition was specified at the rotor outlet. This way, a specified static pressure applies only to the position of minimum radius, which is the rotor hub. The static pressure along the outlet span is calculated from

$$\frac{\partial p}{\partial r} = \rho \frac{u_\theta^2}{r} \quad (2.71)$$

In this expression, r is the distance from the axis of rotation and u_θ is the tangential velocity. Equation (2.71) is valid for negligible radial velocity and negligible streamline curvature.

Periodic boundary conditions were used to model the flow physics in the tip gap, as well as upstream and downstream of the blade surfaces.

When a particle reaches a physical boundary, a boundary condition should be applied in order to get a particular solution of the equation of motion. This could be either impingement of the particle on the wall, or a so-called reflective condition, where the particle normal and tangential momentum before and after the collision with the wall are matched (if the coefficient of restitution is specified to be unity). In the current study, the reflective condition is specified at the solid walls as the boundary condition for the equation of motion of the discrete phase, while particles are allowed to escape at the exit.

2.2.4. Grids

In general, there are several criteria determining the suitability of a grid for a particular flow simulation. These are based, for example, on the turbulence model chosen, convergence, quality of elements, and number of processors used. The selected H-type grid consisted of 132 x 51 x 41 elements in the streamwise, blade-to-blade and the hub-to-tip directions, with two cells in the tip gap. The grid was clustered near the walls using a hyperbolic tangent function, in the areas of rotor inlet and outlet, and adjacent to the leading and trailing edges, where high velocity gradients can be expected during computation. Element equiangle skew, defined as [42]

$$Q_{EAS} = \max \left(\frac{\Theta_{\max} - \Theta_{eq}}{\Theta_{eq}}, \frac{\Theta_{eq} - \Theta_{\min}}{\Theta_{eq}} \right) \quad (2.72)$$

never exceeded 0.8 on the domain, which can easily happen with O-type grids in turbomachinery, especially in the corner regions of the domain of interest. In Equation (2.72), $\Theta_{\max}$ and $\Theta_{\min}$ are the maximum and minimum angles between edges of the element, while Θ_{eq} is the characteristic

angle corresponding to an equilateral cell of a similar shape. Elements with skew higher than 0.9 can cause convergence difficulties during solution. This improved the performance of the numerical simulations, since the flow solver was quite sensitive to elements with high skew. If turbulent dispersion or collision of particles needs to be considered, the problem is even more computationally expensive due to the need to track particle collisions. This task is usually achieved by probabilistic means, like the Monte-Carlo method.

The computational grid can be seen on Figure 2.5, while the designations of the aerodynamic stations are shown in Figure 2.6. Station 1 corresponds to the rotor inlet, while Station 2 to the rotor outlet. Stations LE and TE correspond to surfaces adjacent to the leading and trailing edges respectively.

The same grid has been used for all numerical simulations of the rotor with water injection, except for some validation cases described in Appendix A.

Figure 2.5. NASA Rotor 37 Computational Grid.

Figure 2.6. Meridional View of NASA Rotor 37 with Aerodynamic Survey Locations.

2.2.5. Numerical Methods

2.2.5.1. Solution Procedure for the Equations of the Continuous Phase

The set of conservation equations introduced in Section 2.1.2. can be solved simultaneously with the turbulence model equations in order to determine the flow properties on the domain of interest.

The system of governing equations described by Eq. (2.42) and Eq. (2.43) can be transformed from conserved quantities W to primitive variables Q involving the chain rule:

$$\frac{\partial \vec{W}}{\partial \vec{Q}} \frac{\partial}{\partial t} \int_V \vec{Q} dV + \oint_A (\vec{F} - \vec{G}) \cdot d\vec{A} = \int_V \vec{H} dV \quad (2.73)$$

$$\vec{Q} = \{p \quad u \quad v \quad w \quad T\}^T \quad (2.74)$$

The Jacobian matrix on the left hand side of Eq. (2.73) can be replaced by a pre-conditioning matrix, thus Eq. (2.73) becomes

$$\bar{\bar{\Gamma}} \frac{\partial}{\partial t} \int_V \vec{Q} dV + \oint_A (\vec{F} - \vec{G}) \cdot d\vec{A} = \int_V \vec{H} dV \quad (2.75)$$

Where the preconditioning matrix takes the form

$$\bar{\bar{\Gamma}} = \begin{bmatrix} \Theta & 0 & 0 & 0 & \left. \frac{\partial \rho}{\partial T} \right|_p \\ \Theta u & \rho & 0 & 0 & \left. \frac{\partial \rho}{\partial T} \right|_p u \\ \Theta v & 0 & \rho & 0 & \left. \frac{\partial \rho}{\partial T} \right|_p v \\ \Theta w & 0 & 0 & \rho & \left. \frac{\partial \rho}{\partial T} \right|_p w \\ \Theta H - \delta & \rho & \rho & \rho & \left. \frac{\partial \rho}{\partial T} \right|_p H + \rho c_p \end{bmatrix} \quad (2.76)$$

In the matrix described by Eq. (2.76), θ is chosen in a way that the eigenvalues of the matrix remain well conditioned with respect to time scales. Pre-conditioning has an effect of re-scaling the eigenvalues of the linear system, thus yielding to better convergence characteristics. Flux $\vec{F}$ seen in Eq. (2.75) is evaluated by upwind, flux difference splitting

$$\vec{F} = \frac{1}{2} (\vec{F}_R + \vec{F}_L) - \frac{1}{2} \bar{\bar{\Gamma}} \left(\bar{\bar{M}} \cdot \bar{\bar{\Lambda}} \cdot \bar{\bar{M}}^{-1} \right) (\vec{Q}_R - \vec{Q}_L) \quad (2.77)$$

where $\bar{\bar{\Lambda}}$ is a diagonal matrix of eigenvalues of $\frac{\partial \vec{F}}{\partial \vec{Q}}$; $\bar{\bar{M}}$ is composed of eigenvectors of the

inviscid flux Jacobian. In the implicit scheme, the system of governing equations takes the form

$$\left[\frac{V}{\Delta t} \Gamma + \sum_{j=1}^N S_{j,k} \right] \Delta \vec{Q}^{n+1} = -\vec{R}^n \quad (2.78)$$

where the summation is carried out over the number of faces in the cell. The matrices seen in this expression are:

$$\overline{S}_{j,k} = \frac{\partial \vec{F}_j}{\partial \vec{Q}_k} - \frac{\partial \vec{G}_j}{\partial \vec{Q}_k} \quad (2.79)$$

The residual vector $\vec{R}^n$ is defined as:

$$\vec{R} = \sum_{i=1}^N \left(\vec{F}(\vec{Q}) - \vec{G}(\vec{Q}) \right) \cdot \vec{n}_i A_i - V \vec{H} \quad (2.80)$$

In addition, finally, the time step used for the numerical integration is defined as

$$\Delta t = CFL \frac{\Delta x}{\lambda_{\max}} \quad (2.81)$$

where CFL is the Courant-Friedrichs-Lewy number, $\lambda_{\max}$ is the maximum of local eigenvalues of the inviscid flux Jacobian.

In this investigation, results are obtained using a three-dimensional Reynolds Averaged Navier Stokes Solver, Fluent 6.1. For detailed information, please see the Fluent Users Manual [42].

2.2.5.2. Solution Procedure for the Equations of the Discrete Phase

Once properties of the mixture are known in every computational cell of the domain, the particle trajectory is calculated by means of the integrating the governing equation with respect to

time. During the calculation, the velocity of a droplet is assumed constant while it crosses a computational cell. This way the equation can be modified the following way:

$$\frac{d(\vec{u}_p - \vec{u})}{dt} = -\frac{1}{\tau_v} \rho(\vec{u}_p - \vec{u}) \quad (2.82)$$

Then the analytical solution with appropriate initial conditions takes the following form:

$$(\vec{u}_p^{n+1} - \vec{u}^n) = (\vec{u}_p^n - \vec{u}^n) e^{-\frac{\Delta t}{\tau_v}} \quad (2.83)$$

In the next step, the trajectory of the particle is determined as a function of time by integrating Eq. (2.1):

$$\vec{r} - \vec{r}_0 = \int_0^{\tau} \vec{u}_p dt \quad (2.84)$$

This integration can be carried out numerically using a simple trapezoidal scheme:

$$\vec{r}^{n+1} - \vec{r}^n = \frac{1}{2} \Delta t (\vec{u}_p^n + \vec{u}_p^{n+1}) \quad (2.85)$$

The solver also calculates the temperature and the mass of the droplet by numerically integrating the droplet energy and mass transport equations numerically, according to Eq. (2.31) and Eq. (2.32).

2.2.5.3. Two-way Inter-Phase Coupling Method

This method was developed by Crowe [46] in 1977 and it is only applicable for steady state flow simulations if the volume fraction of the discrete phase is small compared to that of the continuous phase (less than ten percent). The calculation procedure of the solver is as follows:

As a first step of the iteration, initial properties of the flow are calculated in every cell of the computational domain by solving the RANS equations for the mixture, as described in Section 2.2.5.1. If the initial velocity and droplet diameter of the particle are known, the droplet velocity can be calculated by numerically integrating the particle equation of motion with respect to time. This calculation procedure was described in Section 2.2.5.2.

Imagine the computational cell depicted in Figure 2.7. There are N particle trajectories crossing the cell. The water particles are injected into the fluid at a steady rate.

Each particle follows the previous particle injected from the same spot along the calculated trajectory. By definition, the particle number rate along each trajectory can be calculated as

$$\dot{n}_j = \frac{\dot{M}(j)}{\frac{\pi}{6} \rho_d D_0^3} \quad (2.86)$$

where $\dot{M}(j)$ the particle mass flow associated with trajectory j , and D_0 is the initial particle diameter at the place of the injection. This quantity is a measure of the number of particles entering a computational cell on trajectory j per unit time.

Once all trajectories are calculated, the number of particles in each computational cell can be determined by using the relationship

$$n = \sum_{j=1}^N \dot{n}_j \Delta t_j \quad (2.87)$$

where N is the number of trajectories in a computational cell and Δt_j is the time required for a particle to cross the cell on a given trajectory which is labeled by j . The particle volume fraction in the computational cell is determined by the relationship

$$\alpha_D = \frac{\sum_{j=1}^N \dot{n}_j \bar{V}_d \Delta t_j}{V} \quad (2.88)$$

where $\bar{V}_d$ is the average droplet volume in the cell along trajectory j . The volume and mass fraction of the fluid can be easily determined if the particle volume fraction in the computational cell is known, since their sum must be equal to unity. Next, the governing equations of the continuous phase are solved again. In the classical trajectory method, the coupling between the two phases is realized by the incorporation of the heat, momentum and mass gained/lost by every particle in the computational cell. Mass coupling is through the addition of mass through evaporation or by the removal of mass from the computational domain by condensation.

Figure 2.7. Computational Cell with Particle Trajectories.

The mass source for a computational cell is the sum of mass added or subtracted by every droplet residing in the cell:

$$S_{mass} = -\sum_{traj} \dot{n}_j \Delta t_j \cdot \bar{\dot{m}}_j \quad (2.89)$$

If the particles that reside inside the computational cell evaporate, then the quantity $\bar{\dot{m}}_j$, which is the average mass evaporation rate of the particle during its residence in the computational cell along trajectory j , has a negative sign making the overall expression positive.

The momentum exchange between the continuous and discrete phases can be calculated by this expression:

$$S_{mom} = \sum_{traj} \left(\frac{C_D \text{Re}_j}{24\tau_{v,j}} (\vec{u}_{p,j} - \vec{u}) m_j - \vec{u}_{p,j} \dot{m}_j \right) \dot{n}_j \Delta t_j \quad (2.90)$$

The momentum exchange is a result of the drag force acting on the particle and the magnitude of the mass source. If the relative wind is zero (i.e. velocity equilibrium exists) and there is no mass exchange between the phases, then there is no momentum exchange between the phases either. This is why particle relaxation time is such an important similarity number in the characterization of droplet flows. One can think about the second term as a “thrust” similar to one generated by a nozzle on a propulsion system.

Finally, in case of small particle mass loading in the cell, the energy exchange between phases can be calculated as:

$$S_{ener} = -\sum_{traj} \dot{n}_j \Delta t_j \left[\left(h_j A_{p,j} (T - T_p) \right)_j + \dot{m}_j h_s + \frac{C_D \text{Re}_j}{24\tau_{v,j}} (\vec{u}_{p,j} - \vec{u}) \vec{u}_{p,j} \right] \quad (2.91)$$

While the first term in square brackets represents the convective (sensible) heat transfer between phases, the second one is the mass source multiplied by the enthalpy of the carrier fluid at the surface of the particle, accounting for the latent enthalpy added to the mixture by means of

evaporation. The third term represents the work rate of the particle on the surroundings. The expression for energy coupling can also be written in terms of the internal energy of the particles residing in the computational cell:

$$S_{ener} = -\sum_{traj} n \cdot \frac{d \left(m_p c_d T_p + m_p \frac{\|\vec{u}_p\|^2}{2} \right)}{dt} \quad (2.92)$$

Thus, the inter-phase energy source is equal to the energy lost by all droplets in the control volume. If the flow equations represent an ideal fluid consisting of air and vapor, then only the sensible energy exchange will appear as an energy source in the continuous phase equations, but the energy exchange between phases is still represented by Eq. (2.91), since evaporation adds latent heat to the mixture in the form of vapor. The vapor concentration on the domain is determined by solving the species transport equation.

At this point, the three source terms can be assembled into vector $\vec{H}$ on the right hand side of Eq. (2.42) and the equations of the mixture are solved again. This way, the effect of the discrete droplets is incorporated into the continuous phase with every flow iteration. In some cases, the previously described source terms may be relatively large in magnitude, resulting in stability problems (divergence) when integrating the RANS equations numerically. One method to minimize divergence/oscillations is to control the under-relaxation of source terms in vector $\vec{H}$, introducing the impact of the phase change gradually in the RANS equations the following way:

$$\vec{H}_{new} = \vec{H}_{old} + \alpha \left[\vec{H}_{calculated} - \vec{H}_{old} \right] \quad (2.93)$$

When choosing the under-relaxation factor α , one should consider that the particle source terms acting on the fluid phase are proportional to the particle number rate, introduced earlier. This means that the higher the particle rate, and consequently the volume fraction of the discrete phase

on the computational domain, the lower the under-relaxation should be. Another way to accelerate convergence of the multi-phase simulation is to increase the number of flow iterations per discrete phase calculations. The optimal number of flow iterations per discrete phase calculation depends on flow physics.

This way, the two phases are solved alternatively until a converged coupled solution for both phases is achieved. This process is illustrated in Figure 2.8. When one phase is calculated, the other one is always frozen. In the current study, three criteria must be satisfied to assume convergence of the multi-phase flow calculations. These are the following:

- 1) Typically continuous phase residuals should be of order of 10^{-3} or less, and preferably they should oscillate about a mean value and not change with iteration. Sometimes there was difficulty to achieve this condition due to increased oscillation of the residuals around a mean value. This implies that the trajectories of particles do not change with iterations either. Figure 2.9 illustrates the convergence history of the residuals for a typical case.
- 2) Mass flow at the outlet does not change with iteration. The mass flow at inlet and outlet match with accuracy of order 10^{-3} or less.
- 3) Mass source between phases does not change with iterations

Figure 2.8. Schematic Illustration of the Calculation Procedure of the Eulerian-Lagrangian Solver.

Figure 2.9. Residual Convergence History of a Typical Case.

2.3. Computational Model Validation

The quality of the numerical technique used to solve the governing equations should always be evaluated with regard to simple test cases. In order for the solution to be considered valid, at a minimum mass-, momentum- and energy should be conserved through the control volume representing the domain of interest.

In order to validate the two-way coupled Eulerian-Lagrangian multi-phase model used in this investigation, several numerical test cases were performed. In the first case, the objective was to perform a simulation of a transonic compressor rotor test at design conditions without particle injection and compare circumferentially averaged results to experimental ones at the rotor exit. The details of the validation study are discussed in detail in Appendix A.

CHAPTER 3. RESULTS AND ANALYSES

In this chapter, detailed results of simulations of water injection into NASA Rotor 37 are presented.

3.1. Effect of Water Injection on Rotor Performance under Design Conditions

In section, simulations of NASA Rotor 37 with water injection from the inlet are performed at design conditions. Results are then compared to the case without water injection, i.e. dry compression case. This includes comparison of computed circumferentially averaged quantities at the rotor outlet as well as comparison of relative Mach number plots at several spanwise locations; also comparison of rotor operational parameters including mass flow, pressure ratio, and compressor specific work for the each case. Droplet trajectories are also illustrated in order to better understand the interaction of the two phases inside the rotor.

Theoretically, a compressor achieves its highest isentropic efficiency at the design operational point, although sometimes this is not the case, since the stable operating range is equally important when considering all factors that influence design. Consequently, under “ideal” design conditions, one can expect the rotor to operate at its design mass flow and design pressure ratio.

A total of two wet compression cases were performed in this part of the investigation, both at rotor design conditions. Exit static pressure was held at 124325 Pa for all cases at the outlet hub and radial equilibrium conditions were specified along the outlet span. Both

simulations were performed at 100% rotor design speed. Inlet conditions were specified to be standard atmospheric conditions, i.e. total pressure of 101325 Pa, total temperature of 288K, with turbulence intensity of 5% and turbulence length scale 8% of the blade height. The two cases are water droplets over the entire span for 1% total water mass fraction, and a case, which has water droplets injected only in the upper 15% of span at a 0.5% mass fraction.

In the first case, 2000 particle streams were injected from the rotor inlet, normal to the inlet surface. In the second case, 800 particles were injected from the top 15% of the inlet surface. This is meant to simulate a slot of a tip endwall. Each stream originated from a center of an inlet computational cell. The distribution of particle streams was not uniform at the rotor inlet surface; however, particle stream mass flows were weighted to achieve uniform mass flux across every computational cell.

The three cases that will be compared are:

1. Dry case.
2. Wet case with 1% water to air mass flow ratio, uniform inlet injection.
3. Wet case with 0.5% water to air mass flow ratio, injected over upper 15% of inlet span.

Several studies in the past have shown that evaporation of the injected particles is a strong function of initial particle diameter. In this case, it was intended to maximize inter-phase mass and consequently heat exchange inside the rotor, with full evaporation occurring either upstream

or inside of the blade row. This way – in the initial assumption of the author – the cooling brought by the phase change of the water would affect the thermodynamic process in the rotor significantly. Whether this effect would be positive or negative would largely depend on the droplet characteristics at the rotor inlet.

Based on the above reasoning, a constant particle diameter of 1 micron was chosen for the current study. All droplets were injected from the inlet with 30 m/s absolute velocity compared to average flow velocity of 174.8 m/s at that location. Thus, the slip velocity is substantial at the inlet, resulting in intense phase change of particles just downstream of the inlet. The initial temperature for the particles was chosen to be 310 K compared to a fluid total temperature of 288 K at the same location. One advantage of this particle temperature is there is no extra energy needed to heat or cool the fluid, which can be expensive.

Since particles smaller than 5-20 μm in diameter can follow the flow very closely [2], [29], dispersion of particles due to turbulence can be considered negligible. It was also thought that if a small initial particle diameter were chosen then the momentum exchange between phases would be low. In this case, particles will not decrease the axial momentum of the fluid, which could have an adverse effect on the stall margin of the rotor. Injecting with low water to air mass flow ratio also means that one can expect the compressor to operate in a stable range, since rotating stall was observed in a case of water to air mass flows of approximately 3-4% [2].

3.1.1. Uniform Inflow Injection, 1% Water to Air Mass Flow Ratio at Inlet

Figure 3.1 illustrates particle traces in the relative frame of reference with the color depicting the particle diameter. As soon as a particle enters the rotor at the inlet, evaporation

starts to occur due to the difference in vapor molar concentration between the bulk and the droplet surface. The magnitude of inter-phase mass and energy exchange is multiplied in places where there is relative wind present between the droplet and mixture. Due to evaporation, particle diameters decrease downstream until full evaporation of the discrete phase occurs in the rotating blade row. Since the particles follow streamlines very closely, the reflective boundary conditions specified for the discrete phase at walls do not have any physical significance in the particle tracking for this particular case.

Figure 3.1. Particle Trajectories in Rotating Blade Frame of Reference, color represents Particle Diameter.

When the particle equation of motion is integrated to determine its trajectory, the integration time step plays a crucial role in the quality of the numerical solution. Generally speaking, a relatively small particle time step not only means better approximation of spatial and

time derivatives, but also a more stable solution. One way to determine whether the particle time step used is appropriate for the simulations is decreased it in between cases, to see whether droplet trajectories, velocity, computed circumferentially averaged results change as a result.

Figure 3.2 and Figure 3.3 show particle trajectories colored by particle time step, as viewed from the relative frame of reference. If the numerical solution of the particle is conducted, the particle time step used for the integration of the droplet equation of motion is calculated in Fluent according to [42]

$$\Delta t = \frac{L}{u + u_p} \quad (3.1)$$

where L is the length scale set as an input by the user. Smaller value of L essentially means a smaller time step. In Figure 3.2 one can see results for a case when the length scale was set to $10^{-3}m$, while Figure 3.3 shows results for a case when it was decreased by a factor of 10 (i.e. set to $10^{-4}m$). Although the length scale was lowered, the trajectories seem to be unchanged. The same is true for the computed averaged total pressure and temperature ratios at the rotor exit. This means that the original length scale setting yields to solutions that are independent of time step.

Figure 3.4 and Figure 3.5 show the computed relative Mach number contours at mid-span for the dry and 1% wet compression cases discussed at the beginning of the chapter. The leading edge shock is detached in both cases. The flow becomes downstream of the leading edge shock in both cases. However, it is noticeable that this subsonic region is less in the wet compression case, meaning that the shock system is further aft in the wet compression case. When compared with the case without water injection, the relative Mach number decreases in some local areas upstream of the leading edge shock in the wet compression case. The flow continues to accelerate on the suction surface until it reaches the shock from the adjacent blade.

Figure 3.2. Particle Trajectories colored by Tracking Time Step, $L=0.001m$.

Figure 3.3. Particle Trajectories colored by Tracking Time Step, $L=0.0001m$.

At this point, the passage shock interacts with the blade boundary layer, resulting in the smearing of the base of the shock and thickening of the suction side boundary layer downstream. This thickening effect is more pronounced in the wet compression case. The blade wake region has increased as a result of water injection, too. Although this phenomenon can be noticed on the experimental results published by Ulrichs et al. [15], authors do not point it out. This flow feature could be a major source of aerodynamic loss, ultimately leading to the decrease of compressor isentropic efficiency.

Figures 3.5 and 3.6 presents the computed relative Mach number contours at the tip surface corresponding to dry and 1% wet compression cases. For both the dry and wet compression cases, the tip leakage vortex is formed at the blade leading edge due to the strong pressure difference between the blade suction and pressure sides. The passage shock arises due to the deflection of the fluid on the pressure side surface, intersecting the tip leakage vortex in the blade passage. The interaction of the vortex, passage shock and the end-wall boundary layer creates a complicated flow field in this area, the physics of which are not fully understood yet. Considerable progress has been made in understanding this complex interaction due to the advances in computational fluid mechanics. For the case with water injection, the structure of tip flow seems to be slightly different. In this case, the distance between shock waves and the airfoil leading edge seems to decrease. This implies the compressor in the wet case operates further away from stall than in the case without water injection. If this is truly the case, then one can expect that the interaction of the shock and tip leakage vortex is weaker than in the case when no water particles are present in the domain. Time-accurate simulations and experimental observations have shown that as a compressor approaches stall, there is a tendency for the unsteadiness of the tip leakage vortex to increase.

Figure 3.4. Relative Mach Number Contours at 50% Span for Dry Case.

Figure 3.5. Relative Mach Number Contours at 50% Span for Wet Case , $W=1\%$.

Figure 3.6. Relative Mach Number Contours at Tip for Dry Case.

Figure 3.7. Relative Mach Number Contours at Tip for Wet Case, $W=1\%$.

As the compressor is throttled further towards stall, ultimately the tip vortex starts to break up, prompting a low momentum area just downstream of the passage shock. Increasing the back pressure further would widen this area, ultimately producing stall. This low momentum area is clearly distinguishable for the case without water injection, the light-blue area visible on Figure 3.6, which is the part of the tip leakage vortex just downstream of the passage shock. This high blockage producing area seems to have disappeared in the wet case, illustrated on Figure 3.7. This means decreased blockage at the tip, resulting not only in higher entropy production of the tip flows, but also in an operating point that is more stable than in the dry case.

The radial profiles of the circumferentially averaged computational results for the dry and wet 1% cases are compared in Figures 3.8 and 3.9. It is observed that water injection does not seem to affect the calculated total pressure ratio distribution along the span significantly; the pressure ratio of the wet case is slightly lower in the tip area than in the dry case. One possible explanation for this is that both cases were run at the same exit back pressure, corresponding to design conditions.

Water injection across the whole span with 1% water to air mass flow ratio does shift the graph to lower values of total temperature ratio almost uniformly along the whole span. This effect is slightly more pronounced close to the tip. The decrease in circumferentially averaged total temperature at the rotor exit due to water injection does not mean decreased compressor specific work, since, as a result of evaporation the mixture in the rotor exit is composed of two gases: water vapor and air. Because of this, the mixture at the rotor outlet contains not only the enthalpy of the air, but also the enthalpy of the water vapor in latent form. The mixture in the rotor is cooled down by the energy exchange between the two different phases, liquid water and the gas. More precisely, part of the sensible enthalpy of the humid air is converted into latent

Figure 3.8. Comparison of Circumferentially Averaged Total Pressure Ratios for Wet 1% and Dry Compression.

Figure 3.9 Comparison of Circumferentially Averaged Total Temperature Ratios for Wet 1% and Dry Compression.

enthalpy in the form of vapor. This comes at the expense of the increase of relative humidity of the mixture at the exit, as the vapor content of the mixture increases due to evaporating droplets. As one will be able to see, the mass flow through the stage also increases due to the cooling effect, ultimately leading to a higher gas turbine work output.

Figures 3.10 and 3.11 illustrate particle traces in the rotor, colored by the computed relative velocity contours and by slip Reynolds number. According to Figure 3.11, the droplet slip Reynolds number increases in three, clearly identifiable locations of the rotor. These are the area just downstream of the inlet and the relatively large regions just downstream of both the passage and the leading edge shock waves. These are the areas where large velocity gradients are present in the rotor. The particles have certain inertia due to their mass and thus they are unable to closely follow the sudden change in velocity of the continuous phase at the inlet and behind the shocks, or, in fact, of any flow feature that has steep velocity gradients. Particles are injected with an absolute velocity of 30 m/s, while the flow velocity is of order hundred in the inlet region, creating a substantial slip at that location. In zones where trajectories have dark-blue color, the droplets seem to be in velocity equilibrium with the flow. The extent of the velocity non-equilibrium at the shock waves depends on the particle relaxation time, which is a strong function of diameter as seen in Section 2.1. Additionally, the local slip velocity depends on shock strength, which is proportional to the Mach number upstream of it. The stronger the shock, the higher the velocity gradient encountered by the droplet traveling through it, resulting in an increased inter-phase mass, momentum and energy exchange. Thus, evaporation in these zones is much more intense than in other regions, as seen from these results. In addition to decreasing velocity, a droplet traveling through the compressor will experience a sudden increase in static temperature of the fluid downstream of the passage shock. Since the particle has a certain thermal inertia too,

Figure 3.10. Particle Traces in Rotating Frame colored by Relative Flow Velocity Contours.

Figure 3.11. Particle Traces in Rotating Frame Colored by Droplet Slip Reynolds Number Contours.

its temperature will lag behind the temperature of the fluid in a narrow region behind the shock. The result is an increase in the driving potential for convective heat transfer at the particle interface for a very short period of time, resulting in increased energy coupling between the phases, intensifying phase change. This exchange of energy covers much of the energy needed for the phase change of the water particles. The amount of sensible heat extracted this way by the mixture depends on the extent of the temperature non-equilibrium and momentum non-equilibrium. According to Figures 3.10 and 3.11, particles decelerate after reaching the passage shock and fully evaporate just downstream of it.

3.1.2. Injection near the Tip with 0.5% Water to Air Mass Flow Ratio at Inlet

For the case of particles are injected from near the tip at 0.5% water to air mass flow ratio (denoted sometimes in the technical literature by W), the computed particle traces are shown on Figure 3.12, as viewed from the rotating blade frame of reference. The interaction between the two phases, and thus the evaporative cooling effect is now limited to a local area adjacent to the casing end-wall. Due to evaporation, just like in the case with uniform injection of particles across the inlet surface, particle mass decreases downstream, with full evaporation in the blade row, just downstream of the passage shock.

Figure 3.13 shows the particle traces for the tip injection case, colored by particle residence time. The residence time is by definition the amount of time droplet resides in the rotor. This quantity depends on several parameters of the droplet as well as local flow features

Figure 3.12. Particle Traces in Relative Reference Frame Colored by Droplet Mass [kg], Tip Injection Case, $W=0.5\%$.

Figure 3.13. Particle Trajectories Colored by Residence Time [sec], Tip Injection Case, $W=0.5\%$.

encountered by the traversing particles in the rotor. As seen in these results, the maximum residence time of the particles in the rotor is of order 10^{-4} seconds. The short residence time in this particular case can be explained by the relatively small inertia of particles due to their mass and the high velocities encountered in the rotor.

Figures 3.14 and 3.15 compare computed circumferentially averaged flow variables for the dry and wet 0.5% compression cases with tip injection. The computed total pressure ratio in the wet compression case seems to be slightly higher in the tip area compared to the case when no water droplets are present in the rotor. As in the case with uniform injection at the inlet, the computed total temperature ratio decreased for the wet case compared to the dry case, according to Figure 3.15. Interestingly, the cooling of the tip injection seems to affect the upper 40% span, even though the injection was only at the upper 15%. A possible explanation for this phenomenon is that there is radial migration and span-wise mixing of the fluid when it travels downstream in the rotor, especially regions outside of shear layers. Because the particles follow the fluid motion very closely, the cooling effect will also spread to lower radius regions.

Figure 3.14. Comparison of Circumferentially Averaged Total Pressure Ratios for Wet and Dry Compression, Tip Injection Case, $W=0.5\%$.

Figure 3.15. Comparison of Circumferentially Averaged Total Temperature Ratios for Wet and Dry Compression, Tip Injection Case, $W=0.5\%$.

3.1.3. Comparison of Compressor Specific Work for Wet and Dry Cases

The results of the two cases with water injection are briefly summarized and compared to the results of the dry case in Table 3.1. The mass flow at the inlet was calculated by numerical integration of the computed mass flux with respect to area, and the rotor outlet total temperature was calculated by mass averaging.

Table 3.1. NASA Compressor Rotor 37 Computed Results Summary.

	$\frac{\dot{m}_w}{\dot{m}_a}$	$\dot{m}_{inlet} \left[\frac{kg}{s} \right]$	Mass Flow Increase [%]	$T_{t,outlet} [K]$	$w_c \left[\frac{J}{kg} \right]$
Dry	0	19.2	0	369.00	82229
Uniform injection	1	20.16	5	354.85	79989
Tip injection	0.5	20.27	5.6	362.78	81124

As one can see, the mass flow of the air entering the rotor increased in the wet compression cases due to the density increase at the inlet brought by evaporative cooling. In these cases, there is also a sharp drop of total temperature at the outlet, which is the result of the inter-cooling effect of the phase change of particles on the compression. The specific work in the case of uniform injection of water droplets across the inlet was calculated to be 79989 J/kg, compared to 82229 J/kg of the dry compression, which means a 3% decrease with water to air ratio of just 1%. The compressor specific work in both cases was calculated according to

$$w_c = \frac{M_{rotor} \cdot \omega_{rotor}}{\dot{m}} \quad (3.2)$$

where M_{rotor} is the torque acting on the rotor blades, calculated by integration of the pressure with respect to the area of the airfoils and ω_{rotor} is the angular speed of the rotor in radians.

The result of evaluating Eq. (3.2) was found to be equal to the following, calculated with mass averaged quantities in the post-processing phase,

$$w_c = c_p (T_{t2} - T_{t1}) + w_2 h_{fg} \quad (3.3)$$

where w_2 is the vapor to air mass ratio at the rotor exit.

Thus, the compressor specific work is defined with the properties of the (considered ideal gas) mixture, not just the properties of the air. Therefore, one must include the enthalpy of the water vapor content of the mixture at the rotor exit.

The main reason for the decrease of the specific work was found to be the increase of the density upstream and in the rotor due to evaporative cooling, which leads to increased mass flow through the rotor. A 3% reduction in compressor specific work – shown here - does not seem to be significant. In case of multi-stage compressors however, the second stage would compress a mixture of significantly lower temperature. This would mean a reduction of compressor specific work for the second stage, in case the first stage operates under wet compression conditions, provided that mass flow and pressure ratio of the stage stay the same for dry and wet compression. In addition, the mass flow rate of both the compressor and the turbine would increase, leading to a higher net output of the gas turbine. The specific work of the rotor was found to be 81124 J/kg for the tip injection case with $W=0.5\%$, compared to 82229 J/kg of the dry compression, which means a 1.5% decrease. The rise in inlet mass flow due to evaporative cooling is higher in the tip injection case with $W=0.5\%$ compared to the case when water is injected uniformly across the span from the inlet at a higher mass flow rate.

3.2. An Entropy Based Definition of Wet Compression Efficiency

In this section, a method is presented to calculate the isentropic efficiency of the wet compression processes in a transonic compressor rotor for the two cases discussed in Section 3.1, using a second law analysis approach.

3.2.1. Background

In the 1940 and 1950's when there was a surge in the development of aircraft engines, several compressor performance prediction methods appeared in the technical literature, such as those developed by Howell [50]. Broadly speaking all these methods categorized the sources of loss into three main categories, each of them accounting for roughly one-third of the total loss under design conditions. These were profile loss, secondary (induced or end wall) loss and tip leakage loss. Profile loss is usually a loss generated by the boundary layers well away from the end walls, with the assumption the flow is two-dimensional in this region. The trailing edge wake loss is included in the profile loss. End-wall loss is referred as loss due to secondary flows that are generated by the boundary layers close to end walls. Tip leakage loss arises from the clearance of the rotor tip and stator hub. Although there is historical breakdown of loss into the three categories mentioned here, it has been recognized that they are seldom independent. For example, the interaction of the tip leakage and end-wall losses might be quite strong. Entropy does not distinguish between these three loss categories, making the second law analysis a very convenient approach to examine losses in turbomachinery component design.

There are several loss coefficients defined in the technical literature, and for each of them quantities in the relative frame of reference are used. These include the stagnation pressure loss coefficient and the enthalpy loss coefficient. Usually they are proportional to the difference between the thermodynamic properties of an isentropic process to those of a real (irreversible) process or the inverse. All of them reflect some kind of a loss in total pressure due to irreversibility. These coefficients are fine when the relative frame total quantities do not change (i.e. no change in radius). However, in other cases it is inconvenient to use them. An advantage of using entropy in loss calculations is that unlike stagnation pressure and temperature, entropy does not depend on the frame of reference.

Denton [51] states that the only rational measure of loss in an adiabatic machine is entropy creation. Once entropy generation is calculated for every component of the engine, it can be summed up to get the overall efficiency. Denton comes up with an analogy that demonstrates the concept of entropy in a simple way. In his opinion, entropy can be thought of as “smoke”. It can be only generated and advected downstream. Once it has been generated in the compressor, it cannot be destroyed (assuming an adiabatic process). The loss of the machine is then proportional to the “smoke” at the exit of the machine. In his opinion, a more convenient measure of the efficiency is the isentropic efficiency, which is the ratio of the actual compression work to the isentropic (ideal) work. The only factor that can influence the actual work is departure from isentropic flow due to irreversibility or heat transfer. Since flows in compressors are closely adiabatic, for these types of machines only entropy creation by irreversible processes should be considered.

In the past, only a few authors have used the entropy to determine the thermodynamic loss generation in axial compressors. C. S. Tan and P. N. Shah [52] reconstructed the entropy

generation mechanism in a cooled compressor by integrating the viscous dissipation function over the whole domain. These authors integrated to the wall in the post-processing phase. Their solver used a k- ϵ turbulence model with advanced wall treatment in the boundary layers. For their process to work requires that y^+ be less than one for all near-wall cells. Jennions and Adamczyk [53] assess the entropy production of a turbine through the calculation of entropy fluxes through meridional cross-sections of the computational domain. Wall functions were used, but not in the post-processing phase.

In order to calculate the entropy generation rate in a transonic rotor, the analysis of the rotor thermodynamics from the second law perspective is needed. Reynolds Averaged Navier-Stokes solvers use the conservation equations of mass-, momentum- and energy along with the turbulence model equations to analyze a component of the gas turbine. In a second law approach, the primary concern is the availability to perform useful fluid work.

3.2.2. The Second Law of Thermodynamics

The second law of thermodynamics for open systems in steady operation (with one inlet and one outlet, like in turbomachinery) states:

$$\dot{S}_{gen} = \dot{m}_2 s_2 - \dot{m}_1 s_1 - \frac{\dot{Q}}{T} \geq 0 \quad (3.4)$$

where $\dot{m}_1$ and $\dot{m}_2$ are the mass flow through the inlet and exit of the system, s is the specific enthalpy, $\dot{Q}$ is the rate of heat transfer into the system, and T is the temperature of the system boundary. The left hand side of Eq. (3.4) represents the rate of entropy generated within the open system. The first two terms on the right hand side represent entropy flux in and out of the control

volume respectively. Finally, the last term on the right hand side represents the entropy transfer rate due to heat transfer into the system. The second law of thermodynamics states that for any thermodynamic process in open systems the rate of entropy generation on the left hand side is always positive or equal to zero in the reversible limit. The magnitude of $\dot{S}_{gen}$ in this expression shows the extent of the system's departure from reversibility. Since the transonic compressor is considered an adiabatic system and usually mass flow at the outlet is equal to that of the inlet, the second law for compressors simplifies to:

$$\dot{S}_{gen} = \dot{m}(s_2 - s_1) \geq 0 \quad (3.5)$$

While the first law analysis deals with the conservation of the energy in a thermodynamic system and the transformation of energy from one form to another, the second law deals with the availability of energy to do useful work. During any real thermodynamic process some of the potential for energy to perform work is irreversibly lost, which is expressed indirectly by the Gouy-Stodola theorem in thermodynamics. The Gouy-Stodola theorem states that the lost work of a thermodynamic system is directly proportional to the entropy production. In the case of turbines or compressors, the rate of power lost due to irreversibilities is proportional to the rate of entropy generation in the system. Writing the Gouy-Stoloda theorem for compressors yields [54]:

$$h_2 - h_{2,s} = T_0 \dot{S}_{gen} \quad (3.6)$$

In other words, the increase of entropy gives a direct measure of the energy lost in the compressor, energy that is unavailable to do thermodynamic work anymore. Here, T_0 is the absolute temperature of the environment, which in an ideal case is equal to the temperature of the control surfaces surrounding the open system. It can be easily shown using simple trigonometry

on a T-S diagram [51] that the relation between the compressor isentropic efficiency and the change of specific entropy is:

$$\eta_c = 1 - \frac{T_{t2}(s_2 - s_1)}{(h_{t2} - h_{t1})} = 1 - \frac{T_{t2} \cdot \dot{S}_{gen}}{\dot{m}(h_{t2} - h_{t1})} \quad (3.7)$$

Thus, the degradation of isentropic efficiency of the compressor is directly proportional to the increase of specific entropy in the compressor. In other words, in order to optimize the performance of a compressor or any other turbo machine, one needs to minimize the entropy generation rate in the control volume representing the component. Thus, again, entropy as a thermodynamic property seems to be a very convenient way to analyze the performance of a turbomachine.

Another advantage of the entropy-based approach is that it can be applied to rotating turbomachinery components with flow control. As an example, let us consider a compressor rotor with circulation control [55] or another type of device, where a stream of air is blown into the tip region of the rotor in order to extend the operating range of the device.

Figure 3.16. Schematic Representation of a Compressor Rotor with Flow Control.

Alternatively, the jet of air can be blown on the suction side of the device to delay flow separation with increasing angle of incidence [56]. A schematic outline of this kind of a turbomachinery component is shown in Figure 3.16. This system has two inlets and one outlet. For simplicity, assume that the air entering the rotor has a total temperature of $T_{t,0}$, which is the same for the air entering through the flow control device. If the flow control device blows air into the rotor, the integration of the left hand side of the entropy transport equation yields:

$$\dot{S}_{gen} = \dot{m}_2 s_2 - (\dot{m}_1 s_1 + \dot{m}_3 s_3) \quad (3.8)$$

The system should also obey the law of mass conservation, which yields

$$\dot{m}_2 = \dot{m}_1 + \dot{m}_3 \quad (3.9)$$

Once the entropy production rate is known, the isentropic efficiency can be calculated using Eq. (3.7) if the total temperature and entropy of the air entering both inlets is the same. If they are different, the following formula could be applied:

$$\eta_c = 1 - \frac{T_{t2} \dot{S}_{gen}}{\dot{m}_1 (h_{t2} - h_{t1}) + \dot{m}_3 (h_{t2} - h_{t3})} \quad (3.10)$$

The system level understanding of the effect of the flow control is important, but so is an entropy approach to loss.

3.2.3. Derivation of the Entropy Transport Equation for the Mixture

In order to estimate the entropy generation in a transonic compressor rotor one needs to derive a differential equation describing the change of entropy in a differential volume and integrate it through the physical domain of interest. Crowe et al. [41] derived the differential form

of thermal energy equation for compressible flows containing evaporating droplets, by subtracting the mechanical energy equation and the continuity equation from the mixture energy equation:

$$\begin{aligned}
\rho \frac{D(\alpha_c e)}{Dt} = & -p(\bar{\nabla} \cdot (\alpha_c \bar{u})) - p(\bar{\nabla} \cdot (\alpha_d \bar{u}_p)) + \tau_{ik} \frac{\partial(\alpha_c u_i)}{\partial x_k} + \tau_{ik} \frac{\partial(\alpha_d u_{p,i})}{\partial x_k} \\
& + \bar{\nabla} \cdot (k \bar{\nabla} T) + \frac{1}{V} \sum_j h_j A_{s,j} (T_{p,j} - T) - \frac{1}{V} \sum_j \dot{m}_j \left(h_{s,j} + \frac{1}{2} |\bar{u}_{p,j} - \bar{u}_j|^2 \right) \\
& + \frac{1}{V} \sum_j \dot{V}_{p,j} p_{s,j} + \sum_j \frac{3\pi\mu}{V} D_{p,j} \frac{24 \cdot C_{D,j}}{\text{Re}_{p,j}} |\bar{u}_j - \bar{u}_{p,j}|^2
\end{aligned} \tag{3.11}$$

In Equation (3.11), α_c and $\alpha_d = 1 - \alpha_c$ are the volume fraction of the continuous and discrete phases in the control volume, e is the specific internal energy of the mixture, $\dot{V}_{p,j}$ is the rate of change of the volume of individual droplet denoted by index j and $p_{s,j}$ is the pressure of the mixture on the surface of particle j .

The thermal energy equation was derived for a control volume that excluded the droplets, i.e. the control surface surrounding the control volume is at droplet-fluid interfaces, the same way it was done for all derivations of the conservation equations for the mixture in Section 2.1.2. The fluid contained by the control volume was assumed to be the mixture of two ideal gasses, air and water vapor. For further details, please see Crowe et al. [41].

With the assumption that the volume fraction of the droplets is much smaller than unity (i.e., $\alpha_c \gg \alpha_d$) all the terms containing the expression for the discrete phase volume fraction can be neglected. These terms include the flow work due to the motion of droplets, as well as the energy dissipation due to the shear stresses in the continuous phase and particle motion. The work rate associated with the dilatation of the droplets can also be neglected due to the small volume fraction of particles. Since there is low slip Mach number (less or equal than 0.5) on the

domain of interest in the current investigation, terms related to the kinetic energy of particles can also be neglected. Thus, with some manipulation the thermal energy equation of the mixture, Eq. (3.11) can be simplified to yield:

$$\rho \frac{De}{Dt} = -p(\vec{\nabla} \cdot \vec{u}) + \tau_{ik} \frac{\partial u_i}{\partial x_k} + \vec{\nabla} \cdot (k \vec{\nabla} T) + \frac{1}{V} \sum_j h_j A_{s,j} (T_j - T) - \frac{1}{V} \sum_j \dot{m}_j h_{s,j} \quad (3.12)$$

where the summation is over each particle j so the fourth and fifth terms on the right hand side represent the energy coupling between the discrete and continuous phases. The fourth term represents the sensible heat added or subtracted from the mixture by the particles. The last term represents the energy addition by evaporation.

By definition, the specific enthalpy is defined as:

$$h = e + \frac{p}{\rho} \quad (3.13)$$

Substituting the expression for the enthalpy into Eq. (3.12) and using the continuity equation leads to:

$$\rho \frac{Dh}{Dt} = \frac{Dp}{Dt} + \mathbf{T} : \mathbf{D} + \vec{\nabla} \cdot (k \vec{\nabla} T) + \frac{1}{V} \sum_j h_j A_{s,j} (T_j - T) - \frac{1}{V} \sum_j \dot{m}_j h_{s,j} \quad (3.14)$$

Here, $\mathbf{T}$ and $\mathbf{D}$ are shear stress and deformation tensors, respectively. The second term is the double dot product of these, which is a scalar quantity defined the following way, assuming three-dimensional flow and Cartesian tensors:

$$\mathbf{T} : \mathbf{D} = \sum_{i=1}^3 \sum_{j=1}^3 T_{ij} D_{ji} \quad (3.15)$$

The substantial derivative in Equation (3.14) is defined the following way:

$$\frac{D}{Dt}(\) = \frac{\partial}{\partial t}(\) + \vec{u} \cdot \vec{\nabla}(\) \quad (3.16)$$

The differential form of the entropy generation can be expressed as:

$$T \frac{Ds}{Dt} = \frac{Dh}{Dt} - \frac{1}{\rho} \frac{Dp}{Dt} \quad (3.17)$$

Assuming that the sensible heat exchanged by the phases (i.e. the fourth term on the right hand side of Eq. (3.14) is equal to $-\dot{q}_j$, substituting Eq. (3.17) into Eq. (3.14) and rearranging, one can arrive at the differential form of the entropy transport equation for the mixture:

$$\rho \frac{Ds}{Dt} = \vec{\nabla} \cdot \left(\frac{k \vec{\nabla} T}{T} \right) + \frac{\Phi}{T} + \frac{k}{T^2} (\vec{\nabla} T)^2 + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \quad (3.18)$$

Essentially the first and third terms of the right hand side in Eq. (3.18) account for the entropy generation due to the conductive heat transfer to and from the differential element due to mean temperature gradients, since both were derived from the third term on the right hand side of Eq. (3.14). The last term in Eq. (3.18) accounts for the entropy generated by the inter-phase energy exchange. The first term can either increase or decrease the entropy of the system, depending on the direction of the heat transfer. For example, it would be possible to “pump out” entropy from control volume, the way it was done by C. S. Tan and P. N. Shah [52] and thus reducing the overall entropy generation rate of the system. The remaining two terms on the right hand side are positive definite, since the second term is related to the square of the symmetric strain rate tensor while the third term contains the square of the temperature gradient. The third term can play some role in the entropy generation, if relatively large temperature gradients exist in the flow. One can think about this term as a source in the entropy transport equation, provided that a finite temperature difference “triggers it”. In our special case, since flows in turbomachines can be considered closely adiabatic, the first term on right hand side can be neglected [53], because there is no heat transfer through the boundaries. The third term will not be neglected in this study,

although it was found to be smaller than the viscous dissipation term an order of magnitude [53] in a previous investigation.

3.2.4. The Viscous Dissipation Function

The viscous dissipation function is the double dot product of the viscous stress tensor and the velocity gradient tensor. For compressible flow in Cartesian coordinates, it can be expressed as:

$$\Phi = \mathbf{T} : \mathbf{D} = \tau_{ik} \left(\frac{\partial u_i}{\partial x_k} \right) = 2\mu \left(\frac{1}{2} \left(\frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i} \right) - \frac{1}{3} (\vec{\nabla} \cdot \vec{u}) \delta_{ik} \right) \left(\frac{\partial u_i}{\partial x_k} \right) \quad (3.19)$$

Expanding the viscous dissipation function in three-dimensional Cartesian coordinates yields:

$$\begin{aligned} \Phi = \mu \cdot & \left[2 \left(\frac{\partial u}{\partial x} \right)^2 + 2 \left(\frac{\partial v}{\partial y} \right)^2 + 2 \left(\frac{\partial w}{\partial z} \right)^2 + \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right)^2 \right. \\ & \left. + \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 - \frac{2}{3} (\vec{\nabla} \cdot \vec{u})^2 \right] \end{aligned} \quad (3.20)$$

According to Eq. (3.20), the viscous dissipation function is proportional to the squares of velocity gradients at any given position in the physical domain. Divergence of the flow field has some effect on the viscous dissipation function in case of compressible flows, as seen from the last term. From a physical standpoint, Eq. (3.20) shows the dissipation of the kinetic energy of the flow due to irreversibilities caused by friction. Together with Eq. (3.20), Eq. (3.18) shows that the entropy production due to friction will be high in the areas where large velocity gradient coincide with low temperatures.

Multiplying the differential form of the continuity equation by specific entropy yields:

$$s \left(\frac{D\rho}{Dt} + \rho (\bar{\nabla} \cdot \bar{\mathbf{u}}) \right) = 0 \quad (3.21)$$

Adding Eq. (3.18) to Eq. (3.21) together one gets:

$$\frac{D(\rho s)}{Dt} + (\rho s) \bar{\nabla} \cdot \bar{\mathbf{u}} = \frac{\mathbf{T} : \mathbf{D}}{T} + \frac{k}{T^2} (\bar{\nabla} T)^2 + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \quad (3.22)$$

3.2.5. Reynolds Averaged Entropy Transport Equation of the Mixture

Applying the Reynolds-averaging procedure for the right hand side of Eq. (3.22) and time averaging both sides one gets:

$$\overline{\frac{D(\rho s)}{Dt} + (\rho s) \bar{\nabla} \cdot \bar{\mathbf{u}}} = \frac{\overline{(\mathbf{T} + \mathbf{T}') : (\mathbf{D} + \mathbf{D}')}}{\bar{T}} + \frac{k}{\bar{T}^2} \left(\bar{\nabla} (\bar{T} + \bar{T}') \right)^2 + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{\bar{T}} \quad (3.23)$$

This expression further simplifies to:

$$\overline{\frac{D(\rho s)}{Dt} + (\rho s) \bar{\nabla} \cdot \bar{\mathbf{u}}} = \frac{\bar{\mathbf{T}}\mathbf{D} + \bar{\mathbf{T}}'\mathbf{D}'}{\bar{T}} + \frac{k}{\bar{T}^2} \left((\bar{\nabla} \bar{T})^2 + (\bar{\nabla} \bar{T}')^2 \right) + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{\bar{T}} \quad (3.24)$$

For the case of turbulent flows, the time averaged dissipation function, which is the first term on the right hand side, can be decomposed into two components. One accounts for the viscous dissipation due to mean velocity gradients (direct dissipation) while the other accounts for the dissipation as a result of the fluctuating velocity components (turbulent dissipation):

$$\overline{\tau_{ik} \left(\frac{\partial u_i}{\partial x_k} \right)} = \overline{\tau_{ik} \left(\frac{\partial \bar{u}_i}{\partial x_k} \right)} + \overline{\tau'_{ik} \left(\frac{\partial u'_i}{\partial x_k} \right)} \quad (3.25)$$

Expanding further, and writing again in tensor notations:

$$\begin{aligned} \bar{\Phi} = & 2\mu \left(\frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_k} + \frac{\partial \bar{u}_k}{\partial x_i} \right) - \frac{1}{3} (\bar{\nabla} \cdot \bar{\mathbf{u}}) \delta_{ik} \right) \left(\frac{\partial \bar{u}_i}{\partial x_k} \right) \\ & + 2\mu \left(\frac{1}{2} \left(\frac{\partial u'_i}{\partial x_k} + \frac{\partial u'_k}{\partial x_i} \right) - \frac{1}{3} (\bar{\nabla} \cdot \bar{\mathbf{u}}') \delta_{ik} \right) \left(\frac{\partial u'_i}{\partial x_k} \right) \end{aligned} \quad (3.26)$$

The first term of Eq. (3.26) can be easily calculated from the solution of the RANS equations in the post-processing phase. However, the calculation of the second term is not a simple task, especially if the solver does not resolve some regions of turbulent shear layers. There have been several proposed methods for the approximation of this term in the literature [57-59]. Kock et al. [57] recommend calculating the magnitude of the turbulent dissipation using the following equation:

$$\overline{\tau'_{ik} \left(\frac{\partial u'_i}{\partial x_k} \right)} = \rho \varepsilon \quad (3.27)$$

In this expression, the value of ε comes from the two-equation turbulence model. Moore and Moore [60] suggest the following approximation:

$$\frac{1}{T} \overline{\tau'_{ik} \left(\frac{\partial u'_i}{\partial x_k} \right)} = \frac{1}{T} \left(\frac{\mu_T}{\mu_{eff}} \right) \overline{\tau_{ik} \left(\frac{\partial \bar{u}_i}{\partial x_k} \right)} \quad (3.28)$$

The approach that will be used in the current analysis is based on an the idea proposed by Bejan [61], that there should be some kind of an analogy between the viscous dissipation function and the equation for the shear stress for turbulent flows, in a similar way the Reynolds stress tensor is calculated in the RANS equations. The time-averaged viscous dissipation for turbulent flows in pipes then takes the form:

$$\bar{\Phi} = 2\mu_{eff} \left(\frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_k} + \frac{\partial \bar{u}_k}{\partial x_i} \right) - \frac{1}{3} (\bar{\nabla} \cdot \bar{\mathbf{u}}) \delta_{ik} \right) \left(\frac{\partial \bar{u}_i}{\partial x_k} \right) \quad (3.29)$$

Thus, the dissipation due to fluctuating velocity components is readily incorporated with the use of the effective viscosity. Bejan [61] validated the correctness of this hypothesis against experimental results. Analogously, the third term on the right hand side of Eq. (3.24) contains the entropy generation due to heat conduction due to mean temperature gradients and fluctuating ones.

Because of the analogy between momentum and heat transport it seems reasonable to replace this term with one that contains only mean temperature gradients but accounts also for fluctuating temperature gradients, just like it was done by Bejan in Eq. (3.29). Thus, Eq. (3.24) can be reduced to:

$$\overline{\frac{D(\rho s)}{Dt} + (\rho s) \vec{\nabla} \cdot \vec{u}} = \frac{\overline{\Phi}}{\overline{T}} + \frac{k_{eff}}{\overline{T}^2} (\vec{\nabla} \overline{T})^2 + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{\overline{T}} \quad (3.30)$$

The left hand side of this expression is the time average of the entropy production rate in the differential volume containing the mixture.

Assuming steady state, integrating Eq. (3.30) with respect to the volume of the domain of interest and applying the Gauss divergence theorem yields:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \int_{(V)} \left[\frac{\overline{\Phi}}{\overline{T}} + \frac{k_{eff}}{\overline{T}^2} (\vec{\nabla} \overline{T})^2 \right] dV + \int_{(V)} \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{\overline{T}} dV \quad (3.31)$$

For the sake of simplicity, time average notations will not be written out any more. Defining the compressor inlet station as “1” while outlet as “2” yields:

$$\dot{m}_2 s_2 - \dot{m}_1 s_1 = \int_{(V)} \left[\frac{\Phi}{T} + \frac{k}{T^2} (\vec{\nabla} T)^2 \right] dV + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} dV \quad (3.32)$$

Equation (3.32) states that the total entropy generated in a control volume surrounding the compressor rotor per unit time is equal to the volume integral of the irreversibility caused by

viscous friction, conductive heat transfer and energy exchange between phases. This is a very convenient way to examine the loss mechanisms in a compressor stage. Since the viscous dissipation function is proportional to the entropy generation, one can immediately see areas producing high losses in the post-processing phase of a preliminary design, and making the necessary corrections during design iterations. Thus, if one finds the value of the dissipation function per temperature at every location of the physical domain and integrates it with respect to the volume, gets the rate of entropy generation rate of the mixture contained by the rotor. The isentropic efficiency of the compressor rotor then can be calculated using Equation (3.7). There is no need to distinguish between mixing-, trailing edge-, and endwall losses since these are already incorporated in the entropy transport equation.

3.2.6. Entropy Generation in Turbulent Shear Layers

Calculating the integral on the right hand side of Eq. (3.30) in the post-processing phase might lead to erroneous, imprecise results primarily due to high discretization errors in high gradient regions, where mesh might not be fine enough to approximate derivatives with reasonable accuracy. For axial turbomachinery, these local regions include wall boundary layers, wake regions, shock waves, and secondary flows. Since a big portion of losses in transonic compressors are generated in the blade and endwall regions, it is crucial to calculate the entropy generation in these regions as accurately as possible.

Within turbulent boundary layers, the majority of entropy generation occurs within the laminar sub-layer and the logarithmic layer, because the magnitude of the velocity gradient of the flow is the largest in these regions. In the inner part of the turbulent boundary layer, which

contains the laminar sublayer and part of the logarithmic region (about $y^+ < 100$), the velocity gradient is very steep. In fact, according to Denton [51], for turbines, end-wall loss can contribute up to 30% of the total loss, which is a combination of endwall boundary layers and leakage flow losses.

The outer region of the boundary layer, fully dominated by turbulence, does not produce a significant amount of entropy [51], [57] in the case of pipe flows and compressors. Furthermore, the inner regions of the turbulent boundary layers do not seem to be affected by the pressure gradient acting on the flow [51]. In other words, turbulent flow physics are universal in nature, which can be used when calculating the entropy generation in turbulent boundary layers. In order to resolve the velocity distribution in the inner part of the boundary layer properly one would need a very fine grid due to large gradients. This is even true for those computational schemes that resolve the turbulent boundary layer all the way to the wall. F. Kock et al [57] and H. Kawamura [62] have both shown that the entropy generation of heated pipe flow using a $k - \varepsilon$ turbulence model is by far under predicted in the region close to the wall, i.e. $y^+ < 50$. Thus, the prediction of the viscous dissipation function in the inner wall region is a key factor in obtaining accurate results for entropy generation. One needs to integrate the dissipation function in the “core flow” and in the inner parts of the turbulent boundary layers separately and add them together in order to get the total entropy production per unit time.

Several authors have come up with semi-empirical methods to estimate entropy generation in turbulent shear layers. Among them, Bejan [54] derived the local mechanical power dissipation rate per unit pipe length based on the Von-Karman three-segment velocity correlation, for the case of a heated pipe with turbulent flow. F. Kock et al. [57] developed entropy wall functions for incompressible turbulent shear layers based on asymptotic considerations and direct

numerical simulation results of other authors for the same configuration. Their method is usable during the post-processing phase and matches well the data gained by direct numerical simulation by Kawamura. Denton [51] cites a dimensionless dissipation coefficient, defined as:

$$C_{diss} = \frac{T \cdot S}{\rho \cdot V_\delta^3} \quad (3.33)$$

The exact value of this coefficient depends on the state of the boundary layer; however, correlation with experimental work has led to some general results. Schlichting [63] has shown that for turbulent boundary layers the dissipation coefficient is much less dependent on the state of the boundary layer than the more familiar and widely used skin friction coefficient. Denton suggests that for typical turbo machine blades a reasonable approximation seems to be

$$C_{diss} = 0.002 = const \quad (3.34)$$

For laminar boundary layers, the dependency on the boundary layer thickness is noticeable when conducting experiments. Another interesting observation is that in the transition Reynolds number region (where laminar and turbulent flows both could exist), the dissipation in the laminar boundary layer is 2-5 times less than that of the turbulent. This is why it is so important to predict transition in turbomachines.

In the current study, a relationship is developed for the accurate prediction of the total entropy production in turbulent boundary layers next to the wall that will be applied in the post-processing phase. In order to derive the needed equations quantifying entropy generation next to walls, a familiar set of expressions, known as the universal law of wall, is introduced. These are applied along with the k- ϵ turbulence model during simulations.

3.2.7. Wall function treatment

For details about wall functions please see the paper by Shabbir and Turner [64]. As mentioned before, the vast majority of entropy generation occurs in these two layers, the laminar sub-layer and the buffer layer. One can express the average value of the dissipation function across the boundary layer thickness the following way:

$$\Phi_{AVG} = \frac{1}{y_{wall}^+} \int_0^{y_{wall}^+} \Phi dy^+ \quad (3.35)$$

The velocity profiles in the laminar sub-layer and buffer layers, as well as u^* and y_{wall}^+ are known for every computational cell on the wall. Thus, the integral on the right hand side of this expression can be evaluated analytically for each near-wall computational cell.

The velocity profiles in the Reynolds Average Navier Stokes solver (Fluent) used for this investigation were specified in Section 2.2.2. and expressed in physical coordinates:

$$u^+ = y^+ \quad \frac{u}{u^*} = \frac{y \cdot u^*}{\nu} \quad y^+ = \frac{y \cdot u^*}{\nu} \leq 11.225 \quad (3.36)$$

$$u^+ = 2.5 \ln y^+ + 5.5 \quad \frac{u}{u^*} = 2.5 \cdot \ln \frac{y \cdot u^*}{\nu} + 5.5 \quad 11.225 < y^+ \leq y_{wall}^+ \Big|_{wall}$$

where $u^* = \sqrt{\frac{\tau_w}{\rho}}$ is the friction velocity. When $y_{wall}^+ > 11.225$, Eq. (3.36) also can be written the following way:

$$\Phi_{AVG} = \frac{1}{y_{wall}^+} \int_0^{11.225} \Phi(y^+) dy^+ + \frac{1}{y_{wall}^+} \int_{11.225}^{y_{wall}^+} \Phi(y^+) dy \quad (3.37)$$

A change of variable in the integrals on the right hand side from y^+ to y gives:

$$\Phi_{AVG} = \frac{u^*}{y_{wall}^+ \nu} \int_0^{\frac{11.225 \cdot \nu}{u^*}} \tilde{\Phi}(y) dy + \frac{u^*}{y_{wall}^+ \nu} \int_{\frac{11.225 \cdot \nu}{u^*}}^{\frac{y_{wall}^+ \cdot \nu}{u^*}} \tilde{\Phi}(y) dy \quad (3.38)$$

Thus, one can express the average value of the dissipation function across the boundary layer thickness as:

$$\Phi_{AVG} = \frac{u^*}{y_{wall}^+ \cdot \nu} \left(\int_0^{\frac{11.225 \cdot \nu}{u^*}} \mu_{lam} \left(\frac{\partial u}{\partial y} \right)^2 dy + \int_{\frac{11.225 \cdot \nu}{u^*}}^{\frac{y_{wall}^+ \cdot \nu}{u^*}} \mu_{eff} \left(\frac{\partial u}{\partial y} \right)^2 dy \right) \quad (3.39)$$

In this expression, the first term on the right hand side corresponds to the viscous dissipation in the laminar sub layer, while the second term to that in the logarithmic layer. Substituting into Eq. (3.39) the appropriate velocity profiles given by Eq. (3.36) and applying some algebra gives:

$$\Phi_{AVG} = \frac{u^*}{y_{wall}^+ \cdot \nu} \cdot \left[11.225 \mu_{lam} \frac{u^{*3}}{\nu} + 2.5^2 \cdot \mu_{eff} \frac{u^{*4}}{\nu} \left[\frac{y_{wall}^+ - 11.225}{11.225 y_{wall}^+} \right] \right] \quad (3.40)$$

The effective viscosity can be about hundred times the molecular viscosity in the logarithmic layer, coinciding with very high velocity gradients. Thus, one could conclude that the second term in Equation (3.39) would be dominant. This result seems to be consistent with Bejan's [65] findings, who has shown that the dissipation rate in the inner wall region of cylindrical pipes (i.e. $y^+ < 100$) scales the following way:

$$\mathcal{E}_{wall} \sim \frac{u^{*4}}{\nu} \quad (3.41)$$

For the cases when $y_{wall}^+ \leq 11.225$, the average value of the dissipation function across the boundary layer thickness can be calculated as

$$\Phi_{AVG} = \frac{u^*}{y_{wall}^+ \cdot \nu} \cdot y_{wall}^+ \cdot \mu_{lam} \frac{u^{*3}}{\nu} \quad (3.42)$$

Since under the proposed method the dissipation function is evaluated analytically based on the wall functions at the wall used by the CFD solver, it should be more accurate than numerical calculations in general, bypassing errors associated with discretization and machine precision. On the other hand, this method incorporates certain assumptions regarding the state of the boundary layer, which were used in the derivation of wall functions.

Using Eq. (3.31) and Eq. (3.35), it can be shown that the total entropy production in the inner part of the turbulent boundary layer, where most of the entropy generation occurs, can be approximated as:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \int_{(V)} \frac{\Phi}{T} dV \cong \int_{(A)} \frac{\Phi_{AVG}}{T} \cdot \frac{y_{wall}^+ \nu}{u^*} dA \cong \sum_{i=1}^N \left(\frac{\Phi_{AVG}}{T} \right)_i \left(\frac{y_{wall}^+ \nu}{u^*} \right)_i \Delta A_i \quad (3.43)$$

This expression can be evaluated by numerical integration over the wall surfaces during the post-processing phase. In the inner part of the boundary layer, the variation of heat flux due to conduction is quasi-constant and nearly equal to the heat flux at the walls of the rotor, just as the viscous shear is nearly equal to the wall shear stress in this region. Since the walls of the rotor are considered to be adiabatic, it is reasonable to neglect the entropy generation due to conductive heat transfer in this region.

In order to estimate entropy production of the core flow, the right hand side of the entropy transport equation needs to be integrated for the whole domain, except the inner layers of the turbulent boundary layer, as mentioned previously:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \int_{(V)} \left[\frac{\Phi}{T} + \frac{k}{T^2} (\vec{\nabla} T)^2 + \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \right] dV \quad (3.44)$$

One can evaluate Eq. (3.44) numerically the following way:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \sum_{i=1}^M \left[\frac{\Phi_i}{T_i} \Delta V_i + \frac{k_{eff,i}}{T_i^2} (\vec{\nabla} T_i)^2 \Delta V + \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \Delta V_i \right] \quad (3.45)$$

In the core flow of the transonic compressor rotor, relatively high shear occurs in wakes, at the edges of separated regions, shock waves, vortices, and tip leakage. In these areas, turbulent viscosity can be over hundred times the laminar in some cases, which further increases entropy creation. One of the important mixing processes in compressors and turbines is the mixing out of the wake behind a trailing edge. Theoretically, Eq. (3.44) should account for all thermodynamic losses. Since mixing processes obey basic conservation laws, the losses associated with mixing in the core should already be included in the viscous dissipation function, provided that the process is complete by the end of the control volume. The same is true by losses caused by shock waves. In order to minimize the entropy production of the core flow, one would need to minimize the magnitude of the velocity and temperature gradients associated with the terms on the right hand side of Eq. (3.44). The last term of Eq. (3.44) is responsible for the entropy generation due to energy exchange between phases as mentioned before, and will be discussed in detail later.

The mixing loss of the water and airflow is included in the overall entropy generation, which is calculated by the integration of the entropy transport equation, Eq. (3.30). The species distribution of water vapor is governed by a diffusion-convection equation. After this gets solved, results are incorporated into the momentum equation, which are implicit in the entropy transport equation.

The upper limit of integration in Eq. (3.35) is y_{wall}^+ , so evaluating this expression together with Eq. (3.45) would capture the inner regions of the logarithmic layer as well as the laminar sublayer, including not only losses generated by endwall boundary layers but also entropy generated by all shear layers present in this region.

Thus, the rate of entropy production of the mixture in the rotor can be split into two parts:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \dot{S}_I + \dot{S}_{II} \quad (3.46)$$

$$\dot{S}_I = \sum_{V_{core}} \bar{\Phi} \Delta V_i + \sum_{V_{core}} \frac{k_{eff,i}}{T_i^2} \left(\frac{\partial \bar{T}_k}{\partial x_k} \right)^2 \Delta V_i + \sum_{i=1}^N \left(\frac{\Phi_{AVG}}{T} \right)_i \left(\frac{y_{wall}^+ \nu}{u^*} \right)_i \Delta A_i \quad (3.47)$$

$$\dot{S}_{II} = \sum_{core} \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \Delta V \quad (3.48)$$

The first part, $\dot{S}_I$ contains the terms responsible for entropy production due to mean and turbulent viscous dissipation and conduction due to mean temperature gradients. The second part, $\dot{S}_{II}$ contains the terms responsible for the entropy production due to phase coupling. The sum of these can also be evaluated by writing a subroutine in the Fluent RANS solver.

At moderate slip Reynolds numbers and moderate non-equilibrium conditions it is reasonable to assume conditions at the droplet-fluid interface described by the Clausius-Clapeyron relation [14], [45]. Since the Clausius-Clapeyron equations are derived with the assumption of equilibrium between phases, it seems that we can accept this assumption for the current case too. Young [66] states that the condition of equilibrium corresponds to zero entropy production of the mixture. W. Tabakoff et al. [67] assumed zero entropy change of the mixture due to the effect of phase change in their study. The magnitude of entropy generation due to coupling of phases can be readily evaluated in the post-processing phase by numerically integrating the last term of Eq. (3.30) with respect to the volume of the rotor:

$$\sum_{V_{core}} \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \Delta V \quad (3.49)$$

By evaluating this expression, one gets the entropy generation rate of the mixture due to energy exchange between the phases. As mentioned before, the entropy transport equation was derived

for a control volume that ended at the interface of the droplet-liquid flow, i.e. excluded the droplets. A particular form of the droplet energy equation is

$$\frac{d(M_j e_j)}{dt} = \dot{q}_j + \dot{m}_j h_{s,j} \quad (3.50)$$

where e_j is the specific internal energy of particle j , $h_{s,j}$ is the specific enthalpy of the mixture at the particle-mixture interface. This is the quantity of internal energy lost or gained by a single particle. Thus, the entropy release of the particles is equal to:

$$\dot{S}_{particles} = \sum_{V_{core}} \sum_j \frac{\dot{q}_j + \dot{m}_j h_{s,j}}{T} \quad (3.51)$$

Compare this expression with Equation (3.48), i.e.

$$\sum_{V_{core}} \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T_{p,j}} \Delta V \quad (3.52)$$

The overall entropy production of the whole system should include the entropy production of the gaseous mixture and the entropy production of the droplets. Thus, adding together Equation (3.51) and Eq. (3.46) yields to:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \dot{S}_l + \sum_{V_{core}} \frac{1}{V} \sum_j \frac{\dot{q}_j + \dot{m}_j h_{s,j}}{T} \Delta V + \sum_{V_{core}} \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T_{p,j}} \Delta V \quad (3.53)$$

This expression can be further re-arranged:

$$\int_{(A)} \rho s \vec{u} \cdot \vec{n} dA = \dot{S}_l + \sum_{V_{core}} \frac{1}{V} \sum_j \frac{(T_{p,j} - T) [\dot{q}_j + \dot{m}_j h_{s,j}]}{T T_{p,j}} \Delta V \quad (3.54)$$

It is easy to see that the last term, responsible for the entropy production of the droplet and the mixture due to energy exchange, is a strong function of the temperature difference between the phases. This quantity approaches zero when particles are in thermal equilibrium or quasi-equilibrium with the continuous phase. The quantitative evaluation of this term in the post-

processing phase is a difficult task since it contains the average particle temperature in each computational cell, information which is not directly available with the current solver. However, it is possible to approximate this term by assuming an average value for the temperature difference between phases in the rotor. This can be done in the post-processing phase, after determining the temperature difference between mixture and particles, in every position where droplets are present. Taking the average of these may lead to a fair approximation.

3.2.8. Entropy Generation Application for Wet Compression

For the dry and 1% wet compression cases discussed in Section 3.1, the entropy generation rate has been calculated using the proposed approach.

Figures 3.17 and 3.18 show the distribution of computed $\frac{\Phi}{T}$ for the two cases at midspan.

Looking at these results, the advantages of the second law analysis approach on turbomachinery can immediately be noticed. Knowing the value of the dissipation function and temperature at every point of the rotor, one can perform a preliminary assessment about the place where losses get generated and about their magnitude, without further need of post-processing. In axial compressors, areas that produce high losses are the leading edge shock, the blade boundary layers, and the wake behind the trailing edge. Additionally, the passage shock interacts with the suction side boundary layer, resulting in shock smearing at the base and the suction side boundary layer thickening. The entropy generation by the leading edge shock seems to decrease with the effect of water injection according to Figure 3.18. Shock waves are irreversible sources of entropy generation. Normal shock waves tend to produce more entropy than oblique ones, due to high

Figure 3.17. Dissipation Function over Temperature at 50% Span, Dry Compression.

Figure 3.18. Dissipation Function over Temperature at 50% Span for 1% Wet Compression.

viscous normal stresses in the shockwave. The efficiency of the compression process due to a shock of Mach 1.2 is 0.97, which suggests that low Mach-number shock waves are very efficient compression process. The previous conclusion holds until about Mach 1.5, which explains the development of transonic compressors. In this case, although the mass flow of the stage increased due to water injection, the relative Mach number roughly remained the same. The entropy generated by the shock seems to increase compared to the case when water injection is performed. There are indirect sources of loss associated with shockwaves in turbines and compressors. The blade boundary layer and shock wave can form a separation bubble at the foot of the shock. Additional dissipation occurs downstream of this point in the boundary layer, causing extra losses. If the boundary layer is laminar then the probability of the bubble formation is much higher. For a strong normal shock wave, complete boundary layer separation is very likely at Mach number of 1.4. In spite of this, the increment of loss is offset by the pressure increase obtained through the shock wave, making the phenomena perfectly acceptable in case of axial compressors.

Looking at Figure 3.18, it seems that the interaction of the boundary layer on the blade pressure side with the passage shock has increased considerably for the wet compression, which is also true for the pressure side, although the magnitude of increase seems to be lower in this case. A precise formula is developed by Denton to estimate boundary layer loss of airfoils. This formula gives estimate of this loss that is in reality three-dimensional:

$$\xi_s = 2 \sum \frac{C_s}{p \cos \alpha_{ref}} \int_0^1 C_d \left(\frac{V_0}{V_{0,ref}} \right)^3 d \left(\frac{x}{C_s} \right) \quad (3.55)$$

The relative velocity is raised to the third power in this expression. This clearly shows that the suction surface is dominant in producing loss and the regions with high velocity contribute much

higher to the loss. This is substantiated in the current study, where much of the entropy generation occurs on the suction side, and the increase of the entropy generation on the suction surface is by far more dominant than any other factor. The other major contributor of the two dimensional blade losses is the trailing edge loss. For supersonic compressors the mixing loss (entropy generation) behind the trailing edges can account for half of the total two-dimensional loss. With water injection, the entropy generation in the wake region seems to increase too, implying some increase in mixing losses in the wake region.

It is noticeable in Figure 3.17 and Figure 3.18 that in some places there are double lobes in areas corresponding to high loss production. Namely, double lobes appear in the proximity of leading edge shock, which originates from the blade suction surface and also in the wake region just downstream of the airfoil. The viscous dissipation function, Eq. (3.20), can be expressed in a more convenient way as

$$\begin{aligned} \Phi = \frac{1}{2} \mu_{eff} \cdot & \left[\left(2 \frac{\partial u}{\partial x} - \frac{2}{3} (\vec{\nabla} \cdot \vec{u}) \right)^2 + \left(2 \frac{\partial v}{\partial y} - \frac{2}{3} (\vec{\nabla} \cdot \vec{u}) \right)^2 + \left(2 \frac{\partial w}{\partial z} - \frac{2}{3} (\vec{\nabla} \cdot \vec{u}) \right)^2 \right. \\ & \left. + 2 \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right)^2 + 2 \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right)^2 + 2 \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 \right] \end{aligned} \quad (3.56)$$

Since the viscous dissipation function is strongly related to all elements of the derivative tensor, any change in any directional gradient appearing in the flow will have an effect on its value. In other words, one can say that viscous dissipation and loss generation are a three dimensional phenomena. Additionally, areas of low temperature coinciding with gradients can also trigger high entropy generation.

Figure 3.19 shows the plots of effective viscosity, while Figure 3.20 illustrates $\frac{\Phi}{\mu_{eff}}$ at

50% span, for the case without particle injection. The plots seem to contain double lobes, similar to the previous ones.

Figures 3.21, 3.22 and 3.23 show the distribution of the first three terms in Eq. (3.55) at midspan, for the case without water injection. These terms are related to the elements in the main diagonal of the viscous stress tensor. All of these contain double lobes in these areas to a certain extent, indicating that strong directional gradients are present in the flow fields in some areas.

In order to understand the double lobes present in the plots of viscous dissipation function, the relative velocity distribution was plotted along a line on the suction side surface at 50% span, next to the trailing edge, as seen in Fig. 3.24. Figures 3.25 and 3.26 show results of the velocity sampling for the dry and wet 1% cases. The behavior of the fluid in this region is quite complex, since this is the region downstream of where interaction of the suction side boundary layer and the shock occurs. As a consequence, the displacement thickness of the blade boundary layer increases. The relative velocity has three major components, a radial (w_{rad}), a tangential (w_{θ}) and an axial (w_{ax}) component. The relative velocity profile for the dry case is shown in Figure 3.25. In the very inner part of the boundary layer ($y \leq 0.0002$ m), the radial component is more dominant. The fluid loses momentum in this layer, especially in the regions next to the walls, and the centrifugal acceleration acting on it is substantial. Thus, there is radial migration taking place in this region, towards the casing. The radial velocity component decreases towards the outer boundary layer regions, where the axial velocity of the flow is higher, and ultimately switches direction at about $y \approx 0.0028$ m, as seen in Figure 3.25. Since the radial velocity goes to zero at the wall due to the no-slip condition, there is an inflection point present in the radial velocity profile. This creates the deficit in the dissipation function plots, splitting the wake and boundary layer downstream of the blade trailing edge.

Figure 3.19. μ_{eff} at 50% Span, Dry Compression.

Figure 3.20. $\frac{\Phi}{\mu_{eff}}$ at 50% Span, Dry Compression.

Figure 3.21 . $2 \frac{\partial u}{\partial x} - \frac{2}{3} (\vec{\nabla} \cdot \vec{u})$ at 50% Span, Dry Compression.

Figure 3.22. $2 \frac{\partial v}{\partial y} - \frac{2}{3} (\vec{\nabla} \cdot \vec{u})$ at 50% Span, Dry Compression.

Figure 3.23. $2\frac{\partial w}{\partial z} - \frac{2}{3}(\vec{\nabla} \cdot \vec{u})$ at 50% Span, Dry Compression.

Figure 3.24. Velocity Profile Sampling Location.

Figure 3.25. Velocity Distribution in the Boundary Layer on the Blade Suction Side at the Trailing Edge, Dry Case.

Figure 3.26. Velocity Distribution in the Boundary Layer on the Blade Suction Side at the Trailing Edge, Wet Case.

Figure 3.27. Velocity Magnitude ($|\vec{W}|$) Distribution for Dry and Wet Cases.

Figure 3.28. Magnitude of Relative Velocity at Mid-Chord of the Pressure Surface, 50% Span, Wet Case.

For the case of water injection, the speed of sound decreases in the areas adjacent to the rotor inlet due to evaporative cooling, leading to a higher Mach number upstream of the passage shock. This results in increased shock-boundary layer interaction and thus more pronounced thickening of the suction side boundary layer. According to Fig. 3.26, the relative velocity in the region closer to the wall than 0.005 m is almost entirely radial in direction. This also creates an inflection point in the viscous dissipation function, similar to the case without water injection.

Since there is no inter-phase coupling in the tip-injection case in regions that are at 50% or less of the span (Fig. 3.15), the boundary layer-shock interaction in these regions would decrease, resulting in less entropy generation due to viscous dissipation.

Figure 3.27 compares the velocity distribution in the inner part of the turbulent boundary layer at the location depicted in Fig 3.24, for wet and dry cases. The velocity profiles are not classical, which is due to the complex flow physics in this region, especially the radial flows.

Figure 3.28 shows the relative velocity magnitude as a function of distance from the wall, at midspan, on the pressure surface at approximately half chord. Since the flow is more two-dimensional in this region, the profile seems more classical.

Figures 3.29 and 3.30 show computed $\frac{\Phi_{AVG}}{T}$ for the casing end-wall, introduced in Eq. (3.35) for the dry and 0.5% wet cases respectively. In order to calculate the entropy generation in this region of the rotor this quantity needs to be integrated with respect to the area of the wall, as seen in Eq. (3.37). On these plots, there are several, clearly identifiable areas in the tip region that can be related to loss sources. There is a major entropy producing flow feature just downstream of the passage shock, as the result of the complicated interaction of the tip leakage vortex and the passage shock itself. This area seems to widen in the case with water injection, although it is not clear from the plot whether the magnitude of the loss is stronger overall. Theoretically, the closer

the rotor operates to stall, the stronger is the interaction between the shock and tip leakage vortices, which results in this area with high velocity gradient variation and thus viscous dissipation due to the breakdown of the tip leakage vortex. The entropy production in the corner between the blade suction surface and endwall casing seems to increase in the case with water injection. The same seems to be true for the blade wake region, implying an increase in mixing losses in this area. Since this area is close to the wall, substantial migration of low-momentum fluid towards the casing cannot occur. Thus, the double lobe regions in the dissipation plots are not visible in these plots. All these results point toward the conclusion that as a consequence of water injection, the compressor operates off-design with lower isentropic efficiency.

3.2.9. Integrated Results of Entropy Generation

In this part, the entropy generation in several local zones of the rotor was evaluated for the three cases discussed in Section 3.1. All results were calculated by the methodology presented, i.e. each item in Eq. (3.45) has been summed up separately to evaluate the entropy generation rate of the rotor in several zones. For all cases, the entropy production in the inner part of the boundary layers was calculated using Eq. (3.40). Figures 3.31 and 3.32 show the value of y_{wall}^+ for the dry and wet cases with uniform injection. There are certain locations on the suction and tip surfaces where y_{wall}^+ was less than 11.225, which are pointed out by arrows on the plots. In spite of this, the difference between the results of entropy generation calculated with

Figure 3.29. $\frac{\Phi_{AVG}}{T}$ in the Log Portion of the Casing Endwall Boundary Layer for the Dry Case, at Midspan.

Figure 3.30. $\frac{\Phi_{AVG}}{T}$ in the Log Portion of the Casing Endwall Boundary Layer for the 0.5% Wet Case with Tip Injection, at Midspan.

Eq. (3.40) only and Eq. (3.40) and Eq. (3.42) in places where $y_{wall}^+ \leq 11.225$ were found to be of order 10^{-3} or less, making this calculation procedure for this particular case acceptable. Perhaps this is due to the fact that these spots are comparatively small and do not coincide with locations of high friction velocity.

Table 3.2 shows the rate of entropy generation in several local zones of the rotor for the dry, wet and tip injection cases discussed in the previous section. In all cases, nearly 60% of all entropy is generated near the end walls and blade surfaces, which can be explained with the fact that the highest velocity gradients are in the laminar sub-layers and logarithmic layers [65]. Results also show an increase of wet compression entropy production close to other solid walls. One needs to be careful interpreting these results, since for example the entropy generation in the logarithmic layer includes entropy generation rate due to every flow feature next to for the end walls between wall coordinates $0 < y^+ \leq y_{wall}^+$. In case of the tip, for example, this includes the entropy generated by the end wall boundary layer, by the mixing loss of tip leakage flows, shock wave and other features present, not just the classical “profile losses” as the name/characterization of the data in the table might imply. Entropy does not distinguish between the three loss mechanisms so widely used in the design of turbomachinery. The effects of end-wall flow on the loss of compressor blades are dominated by mixing and local separations, which was determined by earlier experiments in previous studies. The interaction of the tip clearance with the end-wall can also have a dominant effect on compressor performance. In the tip injection case, the high-shear producing end-wall area is cooled considerably, thus, even assuming that velocities remain roughly the same, entropy production increases, since entropy production is inversely proportional to the local temperature. The production of viscous dissipation appears to have almost doubled in the wet compression case compared to the dry compression case.

Figure 3.31. Distribution of y^+ along Solid Surfaces, Dry Case (Arrows point to regions where $y^+ \leq 11.225$).

Figure 3.32. Distribution of y^+ along Solid Surfaces, Wet Case (Arrows point to regions where $y^+ \leq 11.225$).

	Dry Case	Wet Case 1%	Wet Case Tip Inj. 0.5%
Laminar Sublayer	1.801	1.771	1.933
Logarithmic Layer	8.811	16.534	14.925
Heat Conduction	0.926	0.799	0.977
Core Flow	5.476	5.435	6.26
S	17.014	24.539	24.095
Calculated η_{is}	0.866	0.811	0.814

Table 3.2. Rate of Entropy Generation $\left[\frac{W}{K}\right]$ and Calculated Isentropic Efficiency of the Rotor for Different Compression Cases.

Looking at Figure 3.18 again, it seems that the effect of thickening of the suction and pressure side boundary layers is somewhat offset by the decrease of entropy generation of the leading edge and passage shocks. In fact, Table 3.2 shows that in the case of wet compression with uniform injection of water particles at the inlet, the entropy production of the core remains almost the same. The entropy generation due to heat conduction seems to decrease for the wet case with uniform particle injection at the inlet. The isentropic efficiencies were calculated based on Equation (3.7). It will later be explained in detail why entropy generation due to energy coupling between phases was not included in the efficiency calculation. Because of the water injection, isentropic efficiencies dropped by almost 5% in both cases. Since the degradation in compressor efficiency is nearly equal for both injection cases, results imply that performance of the rotor is impacted not only by the mass flow of the injected water, but also by the place of the injection. As explained earlier, one of the main effects of water injection is the increased mass flow through the rotor brought by the latent heat of phase change. It should be recalled that the

0.5% tip injection case had a higher increase in mass flow than the 1% wet case. The evaporation of water results in lower loading of the rotor, i.e. higher flow coefficient. This means an off-design compressor operating point. Thus, one could expect a shift from peak to lower efficiencies because of water injection. Meacock et al. [68] showed similar performance degradation by their one-dimensional calculations of a different compressor. Roumeliotis et al. [69] has also achieved similar results in their experimental work, with the assumption of incompressibility during their calculations. Meacock recommended the redesign the compressor in order to make the performance of the stage more favorable under water injection conditions.

Table 3.3 shows the entropy production adjacent to every wall group of the physical domain. As can be seen, the suction side of the blade accounts for almost 50% of the entropy generation in the dry case. For the wet case, the entropy generation in this area almost doubles. Examining Tables 3.2 and 3.3 at the same time tells one that most entropy generation should occur in the logarithmic layer of the suction surface.

In the logarithmic layer the entropy production is proportional to the friction velocity, $\varepsilon_{wall} \sim \frac{u^*{}^4}{\nu}$, as it was shown per Bejan [54], a plot of the square of the magnitude of the wall shear stress was plotted for dry and wet 1% cases and is shown on Figure 3.33 and Figure 3.34. It appears that because of the water injection, there is a relatively narrow area next to the leading edge, where the magnitude of the wall shear stress nearly doubles. This means that the profile loss increased due to water injection. This, together with the temperature drop in the rotor due to evaporative cooling, can explain why entropy is produced extensively in this area. In addition, this seems to be consistent with the fact that the compressor operates off-design, since the angle

of incidence of the blade has changed due to change axial velocity entering the rotor, caused by the change of mixture density.

Zone	Dry Case	Wet Case w/ Tip Inj.
Casing endwalls	1.3925	1.6395
Hub endwalls	2.2309	2.8417
Blade pressure side	2.042	3.4201
Blade suction side	4.948	8.633

Table 3.3. Rate of Entropy Generation [W/K] on Solid Surfaces.

In Section 3.1, it was shown that there is a significant drop of the computed circumferentially averaged total temperature ratio across the span for the case of transonic axial compressors with uniform inlet water injection. For the tip injection case, the same was noticed, but only locally at the tip. The effect of the cooling of the mixture at the tip in this case is more pronounced. Denton [51] showed that entropy generation is likely to be higher in areas of low temperature coinciding with high dissipation. Thus, the increase of entropy production in the wet compression case is most likely due to dissipation at a lower temperature. Alternatively, one can reason that the compressor operates under off-design conditions (“increasing profile loss”), with compression happening at lower temperatures. This also means that the compressor runs at higher corrected speed due to lower inlet temperature, which is the direct result of the evaporative cooling.

Figure 3.33. Square of the Wall Shear Stress on the Blade Suction Surface (View from Casing), Dry Compression.

Figure 3.34. Square of the Wall Shear Stress on the Blade Suction Surface (View from Casing), 1% Wet Compression.

Figure 3.35. Volumetric Rate of Entropy Production $\left[\frac{W}{K \cdot m^3} \right]$ in Zones along the Span.

Figure 3.35 shows the entropy production of several circumferential slices of the rotor across the span. The entropy generation of the 90-100% span region (casing end wall) and the 0-10% span region (hub end wall) add up to almost half of the total entropy generation, due to the complex interaction of the tip leakage vortex, shock wave and end wall boundary layers in this local section. According to Howell [50], annulus loss (i.e. end wall boundary layer loss) and secondary loss (i.e. secondary flows in end wall regions) can cause up to two thirds of total losses in a typical axial compressor. Because of the water injection, results show the entropy production increases across the whole span.

3.2.10. Entropy generation of the continuous phase due to inter-phase coupling

Figure 3.36 shows the entropy generation rate of the mixture in the rotor due to inter-phase energy coupling, as a function of several inlet to air mass flow ratios. This quantity is equal to:

$$\sum_{core} \frac{1}{V} \sum_j \frac{-\dot{q}_j - \dot{m}_j h_{s,j}}{T} \Delta V \quad (3.57)$$

For all cases, the injected flow was uniform across the inlet and the injected particle diameter was 1 μm and the water mass flow was increased from 1.0% to 2.0% with 0.2% increments, with full evaporation occurring in the rotor. Some authors in the technical literature use the expression “entropy change due to phase change”. However, it is preferable to use the term “phase coupling” since energy is transferred from droplets to mixture and vice versa by different modes of heat transfer. As expected, the entropy generation rate of the mixture due to phase coupling seems to be proportional with increasing water injection rates. White et al. [68] simulated wet compression in a compressor using mean-line calculations coupled with evaporation calculations. Authors compared the “aerodynamic” and thermodynamic entropy increase of wet compression, showing that the thermodynamic entropy generation increases rapidly with water injection rate and depending on compression rate it can even exceed the magnitude of aerodynamic entropy generation. Figure 3.36 illustrates that for 1% inlet water to air mass flow ratio the value of the entropy generation due to energy coupling is 20.265 W/K. This seems to be consistent with the findings of White et al., i.e., that aerodynamic and thermodynamic entropy creation rates are of the same order of magnitude. As expected, the rate of entropy generation is proportional with the quantity of water injected into the compressor.

Thus, when injecting water into a compressor, there is entropy generated by the mixture due to inter-phase coupling, and the magnitude of it can be substantial.

Figure 3.36. Rate of Entropy Generation per Volume $\left[\frac{W}{K \cdot m^3} \right]$ of the Mixture due to Inter-Phase Energy Coupling.

3.2.11. Entropy Production of Multiple Phases due to Inter-Phase Coupling

While calculating the overall entropy generation in the rotor, the losses generated by both phases should be taken into consideration, meaning that the entropy productions of the continuous gas and discrete particles should be added together.

Figure 3.37 shows the entropy generation rate of all phases in the rotor due to inter-phase energy coupling as a function of the temperature difference between particles and gas. This

quantity was calculated in the post-processing phase of the simulation. As shown in Section 3.2.7, this expression is equal to the sum of the entropy generated by the mixture and by all the particles residing in the rotor:

$$\sum_{V_{core}} \frac{1}{V} \sum_j \frac{(T_{p,j} - T) [\dot{q}_j + \dot{m}_j h_{s,j}]}{TT_{p,j}} \Delta V \quad (3.58)$$

Figure 3.37 shows that the entropy production of discrete droplets and gas combined is proportional to the extent of thermal-non equilibrium between these. In the current case, on much of the domain the particle temperature is closely equal to the static temperature of the mixture due to a small thermal response time. However, in regions of high gradients the particle temperature lags behind the temperature of the mixture. The average value of the temperature difference between phases was found to be 18.85 degrees Kelvin, which corresponds to 1.25

Figure 3.37. Rate of Entropy Generation per Volume $\left[\frac{W}{K \cdot m^3} \right]$ in the Rotor due to Energy Exchange between Phases.

W/K entropy generation rate. One can think about this quantity as an indicator of the average extent of non-equilibrium of the multi-phase flow in the rotor. The magnitude of this quantity does not seem to be substantial; it is roughly equal to that of the entropy generation of the core flow due to heat conduction. The plot assumes that all particles in the rotor lag behind the temperature of the mixture by this value. Thus, this approximation overestimates the entropy generation of the energy exchange due to phase coupling. In reality, this value should be smaller for the rotor.

3.2.12. Accuracy Assessment

In order to assess the accuracy of the proposed entropy calculation method, the method was validated against the dry results presented in Section 3.1. Using the results of that simulation, mass-flow averaged total pressure and temperature ratio for the rotor were obtained at the inlet and the exit. Entropy cannot be measured directly; it can only be calculated if any two thermodynamic properties of the air are known. For the dry case, since the mixture is assumed to be ideal, it was possible to calculate the specific entropy increase across the rotor using Gibbs equation, namely:

$$\Delta s = c_p \ln \left(\frac{T_{t2}}{T_{t1}} \right) - R \ln \left(\frac{p_{t2}}{p_{t1}} \right) \quad (3.59)$$

The change in specific entropy equals to 29.656 J/kgK for the dry compression case when calculated with the Eq. (3.59). Results obtained by the method presented in this work in the post-processing phase yield a specific entropy generation value of 28.52 J/kgK. Keeping in mind that

the k- ϵ turbulence model might not accurately resolve other flow features with high velocity gradients, this value seems quite accurate.

The correctness of these calculations presented herein is further shown by the fact that by calculating the specific entropy production for the wet compression case with uniform injection with the proposed method yields 42.610 W/kgK and 2.1 W/kgK specific entropy generation rate due to energy exchange between phases. One can also calculate the entropy difference between inlet and exit using the integral of the Gibbs equation and adding entropy change of water and vapor:

$$\Delta s = c_p \ln\left(\frac{T_{t2}}{T_{t1}}\right) - R \ln\left(\frac{p_{t2}}{p_{t1}}\right) + \frac{h_{fg}}{T_{t2}} w_2 - c_d \cdot \ln \frac{T_{t1}}{273} \quad (3.60)$$

Evaluating Equation (3.60) one gets 44.78 W/KgK, which is close.

Thus, the approach based on the integration of the viscous dissipation function seems correct to determine the entropy generation and efficiency of multi-phase flows in axial compressor rotors.

3.3. Parametric Study of Water Injection on Rotor Performance

3.3.1. Effect of Initial Droplet Diameter

This section presents a parametric study of the effect of changing initial droplet diameter on the rotor aerodynamic performance. As a first step, the geometry representing the rotor was scaled up by a factor of four. The reason behind this was to simulate wet compression in a

configuration, which is similar to aero-derivative gas turbine installations in general. This way, all length scales would also increase by a factor of four, prompting an increase in particle residence time, among others.

The circumferentially averaged total temperature ratio profiles seen on Figure 3.38 were calculated for five different cases, containing sprays of constant particle diameters of 5, 10, 15, 20 μm , injected uniformly from the inlet. As particle diameter is increased, one can see a reduction in the cooling effect of phase change across the span, i.e. higher total temperature ratio graphs. The drop in total temperature is the greatest in the case of 5 micron particles, decreasing for larger diameters and becomes the smallest in case of the 20 micron. In all cases, the particles left the rotor downstream. For all cases, 1% water has been injected.

Figure 3.39, Fig. 3.40, Fig. 3.41, and Fig. 3.42 illustrate the relative humidity contours for the four cases at the mid-span section of the rotor.

Figure 3.38. Variation of Total Temperature Ratio with Droplet Size, Uniform 1% Wet Compression Case.

As droplet diameter increases, the relative humidity tends to decrease in the zone, which is downstream of the passage shock, where most evaporation occurs. Once the water changes phase at a droplet surface, the concentration of vapor on the domain is governed by advection and diffusion. The advection happens along streamlines. This means that at the rotor exit one sees the cumulative effect of the phase change that happened in the rotor, in terms of relative humidity.

Summarizing, both relative humidity and circumferentially averaged total temperature ratio plots show decreased evaporation with increasing particle diameters. This can be due to two main reasons. Either the particle residence time in the rotor decreased or the evaporative characteristics of the spray became worse. According to Hamed et al. [26], the temperature increase of the super-cooled particles at the exit of a transonic compressor depend on the time it takes for them to cross the rotor (i.e. particle residence time). Thus, the magnitude of interaction between phases should be proportional to the residence time for a fixed droplet diameter. With the scaling, the inter-phase energy exchange would also increase, making in certain cases evaporative cooling more efficient.

Whenever droplets encounter a steep velocity gradient (shock waves for example), they will attempt to adjust their velocity accordingly, as was already shown in Section 3.1. The larger the particle, the higher its inertia. Thus, it takes longer for a large particle to reach the velocity of the fluid. Consequently, the residence time of the particles in the rotor should depend in a small way on the particle momentum response time. The conclusion that can draw from these results is that the decrease of the evaporative cooling could not happen because of the decrease of particle residence time in the rotor. On the contrary, particles spend longer time in the rotor, as their diameter is increased. The evaporative characteristics of the sprays containing different particle diameters will be examined next.

Figure 3.39. Relative Humidity Contours at 50% Span, $5\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.

Figure 3.40. Relative Humidity Contours at 50% Span, $10\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.

Figure 3.41. Relative Humidity Contours at 50% Span, $15\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.

Figure 3.42. Relative Humidity Contours at 50% Span, $20\mu m$ Droplet Diameter, Uniform 1% Injection across the Span.

The sensible heat exchanged and the mass transfer between the phases is primarily governed by advection and diffusion. The similarity of these transport processes is evident not only at examining the partial differential equations describing these, but also by looking at the expressions of the non-dimensional mass- and heat transfer coefficient for sphere, described by Sherwood and Nusselt correlations for a sphere. Rewriting the equation for mass transfer of the droplet using the definition for Sherwood number of a sphere (Eq. 2.22) and incorporating the expression for particle surface area, the equation for mass transfer between continuous and discrete phases becomes [41]:

$$\frac{dm_p}{dt} = h_c \cdot A_p \cdot \rho \cdot (\omega_{V,\infty} - \omega_{V,r=R_0}) \quad (3.61)$$

Incorporating Eq. (3.61) into the expression of the mass source used in the trajectory method for a computational cell yields to:

$$S_{mass} = - \sum_{k=1}^N h_c \cdot A_{p,k} \cdot \rho \cdot (\omega_{V,\infty} - \omega_{V,r=R_0}) \quad (3.62)$$

The quantity N is equal to the number of particles residing in the cell. It can be calculated as:

$$N = \sum_{traj} \dot{n}_j \Delta t_j \quad (3.63)$$

As one can see, $\sum_{k=1}^N h \cdot A_{p,k}$ has a very important role in the determination of the rate of mass exchange between phases. Equation (3.62) shows that the rate of inter-phase mass exchange is proportional to the area of the droplet-fluid interfaces of all droplets residing in the rotor. Since a big portion of the energy needed for phase change is supplied by the mixture via convective heat transfer, the same is true for the energy coupling between phases. Thus, the finer the spray (i.e. the smaller the droplet diameter of the particles injected into the rotor) the more intense is the

evaporation of the particles resulting in higher cooling effect. The quantity $\sum_{k=1}^N h_c \cdot A_{p,k}$ can be thought of as a kind of mass conductivity of the spray (which is the reciprocal of mass resistance), using the analogy between electrical currents and mass transfer. Similarly, $\sum_{k=1}^N h \cdot A_{p,k}$ is the thermal conductivity of the spray. One needs to examine how these properties changes with droplet diameter. For simplicity, let us now assume that droplets reside in stationary flow and have the same initial diameter. Assuming that one knows the diameter of the droplets contained by the spray and the mass flow of the spray at rotor inlet, the droplet-fluid interfacial area can be calculated as:

$$A_0 = \frac{\dot{m}_p}{m_p} \cdot A_p = \frac{6\dot{m}_p}{\rho_p \cdot D_p} \quad (3.64)$$

Figure 3.43 shows the droplet-fluid interface area of a spray injected into the rotor, with a fixed particle mass fraction of 1%, as a function of droplet diameter. As one can see, the surface area of a spray containing only droplets of 1-micron diameter is 100 times more than the one that contains only 30-micron droplets.

The mass transfer coefficient, h_c , can be determined using the Ranz-Marshall correlation [47]. With the assumption that the relative wind between phases is zero, one gets:

$$h_c = 2 \cdot \frac{D_v}{D} \quad (3.65)$$

Assuming a particle temperature of 310 K, the variation of the mass transfer coefficient with particle diameter can be seen in Figure 3.44. Increasing particle diameter from 5 μm to 20 μm would result in a 75% decrease in the mass transfer coefficient. The same is true for the convective heat transfer coefficient due to the similarity of the two processes.

Figure 3.43. Evaporative Area of Sprays as a Function of Particle Diameter (Injected Mass Flow 0.005 kg/s)

Figure 3.44. Convective Mass Transfer Coefficient as a Function of Particle Diameter (Zero Relative Velocity Assumed).

Thus, the evaporative area of the spray as well as the mass and heat transfer coefficients decrease exponentially with increasing diameter. Therefore, one could see that the mass and thermal conductivity decrease dramatically for sprays with increasing droplet diameters. For a droplet diameter of 30 microns, these are only 1/3000 of the original quantity, corresponding to droplet diameter of 1 micron. Thus, the finer the spray, the better the evaporative effect, mainly due to the decrease of the mass- and thermal resistance of individual particles.

3.3.2. Effect of the Injection of a Distribution of Particles

Usually, a nozzle provides a spray containing droplets of several diameters. As it was already shown, the finer the spray, the better the evaporative effect. Thus, the main objective of nozzle design for the objective of evaporative cooling should be the achievement of as fine spray as possible. Besides, the negative effects of evaporative cooling, outlined in the literature review and introduction sections are minimized this way. Usually, the droplets exiting the nozzle are described by a statistical mean. The “quality” of distribution of particles is influenced by several factors, including nozzle shape, nozzle supply pressure, and water quality. In the industry, usually two types of distributions are used to characterize nozzles: mass distribution and number distribution of particles.

The mass distribution of particles is generally described by Rosin-Rammler or Rosin-Rammler logarithmic distribution functions [41]. If $f(D_p)$ is the number frequency distribution of a spray (probability density function), then the cumulative distribution function by definition is equal to:

$$P(\xi \leq D_p) = \int_{-\infty}^x f(D_p) dD_p \quad (3.66)$$

Since the Rosin-Rammler distribution is a two-parameter distribution, it is relatively flexible to describe a set of samples, thus, it provides a convenient way to describe spray characteristics. By definition, it is equal to:

$$P(\xi \leq D_p) = 1 - e^{-\left(\frac{D_p}{\bar{D}}\right)^N} \quad (3.67)$$

Equation (3.66) shows the probability that the diameter of a random particle in the spray (for example, if taken by sampling), is smaller or equal to the droplet of diameter D . If the Rosin-Rammler distribution function was gained by integration of a mass frequency distribution function, then Equation (3.66) shows us how much mass fraction of the particle spray is associated with particle sizes less than D . The Rosin-Rammler distribution has two parameters: $\bar{D}$ is called the size parameter in the distribution and the parameter N is called spread parameter. The spread parameter is proportional to the variance of the distribution.

For the purpose of this study, droplets were injected from the inlet portion adjacent to the casing, as described in Section 3.1. The distribution of particles was described by Rosin-Rammler logarithmic distribution, with the following diameters and parameters: minimum particle diameter of $1\mu m$, maximum particle diameter of $11\mu m$, size parameter of $5\mu m$, spread diameter of 1.1 and number of particle sizes 6. Particle temperature at the inlet was set to $330K$ with water to air mass flow ratio of 0.4%.

The cumulative mass distribution function of the injected spray is shown on Figure 3.45. The points seen on the graph correspond to the particle sizes used to define the injection. 60% of

the total mass flow of the spray corresponds to particle diameters 5 microns or less, which essentially favors evaporation.

In the technical literature the most frequently used diameter to characterize the spray of particles has been the Sauter Mean Diameter, which represents a hypothetical droplet, whose ratio of volume to surface area is equal to that of the entire spray [2]. This parameter is very useful to characterize evaporative cooling potential in some applications, since it deals with surface area, which, as it was shown, is a crucial factor in the mass and energy exchange between phases. The Sauter Mean Diameter is defined as:

$$D_{SMD} = \frac{\int_0^{\infty} D^3 f(D) dD}{\int_0^{\infty} D^2 f(D) dD} \quad (3.68)$$

If a spray only contains particles of one certain diameter D_p , then the value of the Sauter Mean Diameter is equal to D_p . This parameter, however, might not characterize the spray appropriately for the current application, since the effect of droplet diameter not only influences mass and heat transfer exchange processes, but also on momentum transfer between phases, for which Sauter Mean Diameter does not account for. A more appropriate measure would be to characterize the spray by calculating the mean diameter of the distribution, which corresponds to the expected value of the distribution of the spray:

$$\mu = \int_0^{\infty} D f(D) dD \quad (3.69)$$

In order to verify this hypothesis, a run with constant particle diameter equal to the expected value of the distribution (also called overall mean diameter), namely $4.54 \mu m$ was also performed. Other parameters like particle temperature and velocity at the inlet were the same as

in the other case. Circumferentially averaged results were then compared to one another for the two runs, in order to assess the aerodynamic performance of the rotor in the case when particles of several diameters are injected into the inlet simultaneously.

As one can see from Figures 3.46, Fig. 3.47 and Fig. 3.48, computed circumferentially averaged quantities and shock position line up fairly well for cases with constant particle diameter and when the distribution of particles is injected. Thus, the effect of an injection of particles following the above-described distribution is roughly equivalent to an injection with a constant particle diameter of $4.54\mu m$ for inter-phase mass-, momentum- and energy exchange perspective.

Figure 3.45. Cumulative Distribution Function of Injected Spray.

Figure 3.46. Comparison of Total Pressure Ratio Profiles for the cases.

Figure 3.47. Comparison of Total Temperature Ratio Profiles.

Figure 3.48. Averaged Static Pressure as a Function of Axial Coordinate at Casing.

3.3.3. Effect of Initial Droplet Temperature

Figure 3.49 illustrates the circumferentially averaged total temperature ratio profiles along the span for the dry compression case and 1% wet compression cases with uniform injection of droplets at the inlet with initial particle temperatures 300 K, 320 K and 340 K respectively.

According to Chaker et al. [2], the increase of initial particle temperature does not influence the speed of evaporation in a significant way, although it does increase the vapor pressure at the liquid-vapor interface of the particle. Thus, evaporation does not occur significantly faster. In this case, particles will quickly cool down after they enter the rotor, since

the thermal response time is small. Thus, evaporation will intensify only for a very short period of time, the increasing the convective heat transfer from fluid to droplet.

Appendix B contains a detailed derivation of the expression of the magnitude of the inter-phase energy exchange with droplet initial parameters. Rearranging the final result of Appendix B, one gets the following expression for the sensible heat exchange between mixture and particles, when particles evaporate fully in the rotor:

$$q = (m_p c_p T_p)_1 - (m_p)_1 h_s \quad (3.70)$$

Since all the variables seen on the right hand side are known from the simulation, the decrease in sensible enthalpy due to water injection can be calculated. Essentially the drop in circumferentially averaged total temperature ratio across the rotor will be proportional to this.

According to Eq. (3.70), injected water droplets with higher temperature will initially have higher internal energy compared to those injected at lower temperature. As it was shown before, the heat exchanged with the continuous phase by convection and the energy required for phase change balances the internal energy of the droplet. If particles do not evaporate at wet bulb temperature, a large portion of energy needed for phase change decreases the internal energy of the particle (i.e. cooling the particle) instead of extracting the energy from the mixture. One can think about the particle as a storage of internal energy that can be used for several purposes.

Thus, one can conclude that for wet compression the lower the original temperature of the fluid, the better the cooling effect, in cases when full evaporation occurs inside the rotor. This is why increasing droplet initial temperature will result in less effective evaporative cooling, which manifests itself in Figure 3.49 in a total temperature ratio increase.

Figure 3.49. Effect of Droplet Initial Temperature on the Total Temperature Ratio Profile.

3.3.4. Effect of Water to Air Mass Flow

The relative humidity of an air-water mixture is defined as the ratio of the partial pressure of water vapor in the mixture to the saturated vapor pressure of water at a given temperature:

$$\phi = \frac{p'_{vap}}{p_{sat}(T)} \quad (3.71)$$

Theoretically, under normal conditions, ϕ cannot exceed 100%, unless a mixture is supersaturated.

Figure 3.50 shows the relative humidity at 50% span of the rotor for an injected water to air ratio of 1%. The maximum value of the plot is located between leading edge and passage

shocks. Since ϕ never exceeds 60% in the rotor, overspray or condensation conditions – which the current solver cannot model correctly – do not exist. In the next step, the injected water to air ratio was increased from 1.0% to 1.4%. A particular concern in the industry with increasing the mass flow of injected water has been compressor surge. According to Chaker et al. [2] experience has shown that no surge related problems have been reported for over fogging levels as high as 0.6-1.0 % mass flow.

Figure 3.51 shows the relative humidity at 50% span of the rotor for this case. As it was seen in the previous case, there is a sharp increase in ϕ right downstream of the leading edge shock. The main reason behind this seems to be a sudden decrease of mixture static temperature at this place, lowering the saturation pressure of water vapor.

The opposite happens downstream of the passage shock, this time due to the increase of static temperature. In this case, the mixture approaches the fully saturated state between the two shocks, where the relative velocity of the fluid is supersonic. If the mass flow of the injected water was to be increased further, condensation could occur between the leading edge and passage shock. This implies the existence of a limit of water to air ratio for the injection that has the purpose of performance improvement. Increasing the injected water mass flow beyond this point would make full evaporation impossible for the rotor, even for the case of small particles. Additionally, negative effects such as erosion would increase in the rotor. Thus, in order to perform a wash of transonic compressor stages (often used to gain efficiency by cleaning the compressor), water particle size needs to be greater than a specific value or certain mass flow; otherwise, one can expect evaporation to occur in the rotating blade row. One interesting area to explore in the future would be the determination of a “limit diameter” of particles above which full evaporation is not possible for the current conditions at the lambda shock. In the current

Figure 3.50. Relative Humidity Contours at 50% Span for Water to Air Mass Flow Ratio of 1%.

Figure 3.51. Relative Humidity Contours at 50% Span for Water to Air Mass Flow Ratio of 1.4%.

Figure 3.52. Total Temperature Ratio Profiles Corresponding to Different Injected Water to Air Ratios.

study of the effect of particle diameter for a four times larger configuration, all particles greater than 2 microns in diameter did not fully evaporate inside the rotor.

Figure 3.52 shows the total temperature ratio profiles for injected water to air mass flow ratio of 1.0% and 1.4%. As one can expect, increasing the water to air ratio leads to increased evaporative cooling effect. This can also be seen from the thermal energy equation of the mixture containing droplets. In this case, more droplets of the same diameter and thus, specific evaporative area enter the compressor rotor. The total evaporative area, however, increases proportionally with the water mass flow. This is represented in a further shift of total temperature ratio plot to lower values, meaning higher mass flow through the rotor. By increasing only the mass flow of injected water, the effective evaporative area was increased, since droplet diameter

remained the same. This is why one can see increasing cooling effect, which is proportional to phase change.

3.4. Off-design Performance with Water Injection

One of the current development trends in axial compressor design is the increase in stage loading, which in turn tends to decrease the operating range of the turbomachine. The role of the tip flows in the high-speed compressor stall was shown to be crucial by several authors [70-73]. As the static pressure is increased at the hub of the rotor exit, the blockage increases rapidly, especially next to the casing end wall. The blockage effect leads to higher incidence angles at the tip, throttling that rotor toward stall.

Previous studies have shown that the pressure ratio and operating range of a compressor stage can be significantly increased by controlling the development of the airfoil and end-wall boundary layers.

In this part of the study, an attempt was made to determine the rotor speed lines with water injection and compare them with the dry compression speedline. This essentially includes the determination of the stall and choke points for all cases. In addition, the mechanisms affecting the change of stable operational range of the compressor numerical stall in case of evaporating water particles present are examined in detail.

3.4.1. Shock Position for the Dry and Wet 0.5% Cases

Figure 3.53 shows the circumferentially averaged static pressure distribution at the tip, along the axial coordinate of the rotor for two cases. In one case, the back pressure at the exit hub was held at 23000 Pa gauge and no particles injected. This corresponds to design operating point of the rotor. In the other case, the pressure at the rotor exit hub was set to 25000 Pa gauge and water injected from the upper 20% of the span from the inlet, as described earlier. The plots line up well when the back pressure at the hub is 25000 Pa gauge, meaning that with the tip injection one has a reserve of 2000 Pa gauge back pressure before reaching the same shock position, implying an improvement in compressor stability for this case.

Figure 3.54 shows the circumferentially averaged static pressure distribution at the tip along the axial coordinate of the rotor for four cases. These are: dry compression, a case with uniform injection across the span with particle diameter 1 micron with 1% water to air ratio, tip injection with 1 micron and tip injection with 5-micron particle sizes with 0.5% water to air ratios. Again, the jump in static pressure on the plot corresponds to passage shock location at the tip. This plot shows that both injections with a particle diameter of $1\mu m$ shift the graph to the right, meaning that in both wet compression cases the shock moved downstream, which, again, could also be a sign of compressor operating point moving away from stall [74]. This effect seems slightly more pronounced in the case of tip injection with 1 micron particles, as seen from this figure, despite of the fact that the water to air mass flow ratio at the inlet for the second case decreased by half and the place of injection changed.

Thus, injecting water particle streams from a slot close to the tip seems to be an effective way to control compressor stability along with the benefit of mass flow increase, which yields higher net power output from the gas turbine installation.

Figure 3.53. Averaged Static Pressure as a Function of Axial Coordinate at Casing, Throttling Case.

Figure 3.54. Averaged Static Pressure as a Function of Axial Coordinate at Casing.

On the contrary, when particles of $5\ \mu\text{m}$ in diameter are injected at the tip, the corresponding static pressure jump seems to move upstream, potentially pushing the compressor towards stall.

3.4.2. Change in the Axial Momentum of Tip Flows due to Water Injection

Figure 3.55 shows the variation of the axial momentum of the fluid at the rotor outlet as function of span, for three wet compression cases containing particle diameters of $1\ \mu\text{m}$, $3\ \mu\text{m}$ and $5\ \mu\text{m}$, with 0.5% water to air ratio. The momentum of the flow is defined as ρu .

In all cases, particles were injected near the tip at the inlet in the 20% of span that occupied approximately 20% of the span. This way, it was assumed that evaporative cooling would have a certain effect on the compressor rotor stability.

As a result of tip injection, the axial momentum seems to change across the whole span for all cases, but especially in the region close to the tip, i.e. the upper 15% of the span. In the case of 1 micron particles the axial momentum at the tip increases, while for those particles where full evaporation does not occur in the rotor (3 and 5 microns) axial momentum of the fluid decreases. Thus, this is yet another proof that injecting fluid into the low-momentum tip flows, responsible for up to 30% of aerodynamic losses in axial compressors [51], will have an effect on compressor operational stability. If this effect is positive, the increase of axial momentum close to the tip can also have a positive impact on the stall margin of the downstream stages [75], [76] as well as the current stage.

Local change in axial momentum can be a result of two things if evaporating droplets are present:

- 1) Increase of mass flux through the rotor, which would be the direct result of evaporative cooling.
- 2) Increase of the magnitude of inter-phase momentum coupling, i.e. of the interaction between fluid and droplets, due to the drag force acting on the later ones.

Table 3.4 shows the volume integrals of the inter-phase volumetric mass and momentum sources for both cases, evaluated numerically in the post-processing phase of the simulation. These quantities can be expressed the following way. The integral of the mass source is equal to:

$$\int_{(V)} - \frac{\left(\sum_{i=1}^N \dot{m}_{P,i} \right)}{V} dV \quad (3.72)$$

Figure 3.55. Axial Momentum as a Function of Injected Particle Diameter, Tip Injection Case.

Essentially this quantity is an indicator of how much mass is exchanged between phases in the rotor per unit time. The integral of the momentum source is essentially has the same format, but instead of the volumetric mass, the momentum source appears in the integral.

According to Table 3.4, the volume integral of the mass source is equal to 0.00063 kg/s for the case when particles of $5\mu m$ in diameter are injected into the rotor, while it is substantially higher, 0.002 kg/s for the $1\mu m$ case. The volume integral of the axial momentum source term is positive +0.063 N for the $1\mu m$ case, versus -0.19 N for the $5\mu m$ micron case.

D_p	Mass Source	Momentum Source
1 micron	0.002 kg/s	+0.063 N
5 micron	0.00063 kg/s	-0.1907 N

Table 3.4. Inter-Phase Mass- and Momentum Sources for Different Injection Cases.

In other words, these results indicate that particles of $1\mu m$ loose more mass due to evaporation in the rotor than particles that are larger in diameter. That implies more energy extracted from the compressor in order for the phase change to happen. This essentially means higher mixture density in the rotor than in the $5\mu m$ case. This must be due to the decrease of evaporative area of the spray consisting of $5\mu m$ particles, compared to droplets with a diameter of $1\mu m$. In case of 1-micron case, much of evaporation seems to happen upstream and inside the rotor, as we have seen in Section 3.1. These places are the inlet and the shock waves, where large flow gradients

are present. On the contrary, 5-micron particles reach the rotor exit without much evaporation happening in the rotor.

When the value of the momentum source is negative, the particles tend to decrease the axial momentum of the mixture. This seems to be the case for the injection with 5-micron particles. The momentum source is negative and of order of magnitude 10^{-1} N.

Thus, the main reason for the axial momentum increase in the 1-micron case seems to be the increase of density of the mixture entering into the rotor, with contribution from the evaporating water particles. Additionally, the positive momentum source seen in this case contributes to this effect too. The higher the diameter of the injected particles, the lower seems to be the increase of density in the rotor due to cooling. Additionally, in these cases particles tend to decelerate the flow, throttling the rotor towards stall.

The surface integral of the axial momentum for a given control volume is equal to:

$$\frac{\int_A \rho u \vec{u} \cdot \vec{n} dA}{\int_A \rho \vec{u} \cdot \vec{n} dA} \cdot \dot{m} \quad (3.73)$$

This quantity can be derived by integrating the left hand side of the steady axial momentum equation in Cartesian coordinates with respect to a control volume and applying the Green's theorem.

We will now examine the axial momentum in the upper tip endwall region for both tip injection cases, at several surfaces corresponding to measurement locations. The upper end wall region is a zone next to the casing with a height of roughly ten times the tip clearance, where interaction of the end wall boundary layer, shock and tip vortex produce significant losses.

Table 3.5 presents the variation of the overall axial momentum at several rotor locations

for the two tip injection cases, namely $1\mu m$ particles and $5\mu m$ particles, as well as the volume integral of the inter-phase axial momentum source in the rotor (which appeared in the momentum equation of the mixture).

The surface integral of the axial momentum seems to be the highest for the case when particles of $1\mu m$ diameter are injected. This is noticeable at every location, especially at the trailing edge and downstream, after the full evaporation of particles has already occurred. The volume integral of axial momentum is low in this case. Additionally, its sign is positive, meaning that particles add to the mixture momentum.

The volume integral of the axial momentum source is highest for the $5\mu m$ case according to the table and it has a negative sign, meaning that particles decelerate the flow.

Additionally, since there is no significant evaporative cooling happening in the $5\mu m$ case, the axial momentum is a lot lower in almost every location due to the significant axial momentum source in this region of the flow.

D_p	1	LE	TE	2	Volume Integral of the Axial Momentum Source
Dry	21.318	26.514	20.896	25.974	0
$1\mu m$	21.7286	26.7696	21.682	26.799	+0.063
$5\mu m$	19.137	24.596	17.798	21.677	-0.1907

Table 3.5. Variation of Axial Momentum at Several Rotor Aerodynamic Locations.

The increase in magnitude of the inter-phase momentum exchange occurring downstream of the rotor entrance seems to affect the flow at the inlet, in this case in a negative way. Since the conditions at the rotor inlet are subsonic, the velocity of the mixture at the entrance is directly extrapolated from the interior, which seems to have lower momentum in this case due to the inter-phase energy coupling.

Ulrichs and Joos [15] have noticed in their experimental investigation of a similar compressor cascade the decrease of velocity around the airfoil due to the interaction of higher inertia particles with the flow. They attributed this phenomena to the fact that for high diameter particles it takes longer time to reach the speed of the flow (i.e. higher relaxation time of particles with greater diameter).

By definition, the slip Reynolds number of the droplet is directly proportional to the inertial forces acting on it. Because of this, one useful way to get a feel for the magnitude of the inter-phase momentum exchange locally in the flow field is to plot particle trajectories colored by particle Reynolds number based on relative wind.

Figures 3.56 and 3.57 show the particle trajectories in the rotor, for the cases of tip injection of constant $1\mu m$ and $5\mu m$ particles, respectively. On these plots, droplet paths are colored by Reynolds number based on relative wind. Results are similar to the case with uniform injection at the inlet, the slip Reynolds number is larger than zero at the inlet in areas where large gradients are present, i.e., at the rotor inlet and the leading edge and passage shocks. However, for the 5-micron case, the slip Reynolds number seems to be non-zero in a significantly wider portion of the domain, compared to the 1-micron case. This essentially means that in the first case, droplets are more sensitive to change in fluid velocity. Since the momentum and thermal response time is proportional to particle diameter, the greater the particle diameter, the higher the

Figure 3.56. Particle Traces Colored by Slip Reynolds Number, $1\mu m$ Case.

Figure 3.57. Particle Traces Colored by Slip Reynolds Number, $5\mu m$ Case.

degree of momentum non-equilibrium between particle and fluid at places where high velocity gradients are present. The same is true for the extent of thermal non-equilibrium at places where droplet traverses high temperature gradients. This should lead to large contribution of the drag force of the particles to the momentum source.

Figures 3.58 and 3.59 show particle paths colored by residence time for the two cases. As one can see, the residence time increases substantially for the case with larger diameter. This is due to the increase in momentum response time. As a result, the interaction time between the different phases increases.

Thus, one can conclude that the increase of axial momentum should be primarily due to the mass and energy transfer between the phases, i.e. increased density at the inlet and downstream. Significant difference in axial momentum between the cases seems to occur after the leading edge. However, for the cases when particles of larger diameter are injected, negative effect on the momentum is more pronounced and noticeable even at the rotor inlet. It seems that increasing particle diameter of the spray is equivalent to throttling the rotor.

3.4.3. Computed Speedlines for Dry and Wet Compression Cases

In the next step, the compressor characteristics were calculated for two cases. At first, droplets were injected uniformly from the rotor inlet, accounting for 1% of the flow. In the second case, water droplets were injected in the top 20% of span accounting for 0.5% of the flow. In both cases, the droplets were 1 micron in diameter, at a temperature of 310 K and velocity of 30 m/s. During the process, the back pressure at the hub of the rotor exit was slowly

Figure 3.58. Particle Residence Time [s], Tip Injection with $1\mu m$ Particles.

Figure 3.59. Particle Residence Time, Tip Injection with $5\mu m$ particles

raised from 18000 Pa gauge by 1500 Pascal increments, until a point where converged solution was no longer possible. This is the operating point corresponding to the numerical stall, a phenomena described by Jennions and Turner [74]. When the back pressure at the rotor exit hub is raised, both the leading edge shock and the tip vortex are pushed gradually upstream until they intersected the rotor inlet boundary. Increasing the rotor back pressure further lends to reversed flow at the tip and the inability to achieve a converged numerical solution. As the rotor is throttled closer to stall, the increasing shock-vortex-endwall boundary layer interaction near the casing leads to vortex breakup. As a result, blockage in the tip region increases, resulting in oscillating flow residuals during the numerical simulation and ultimately producing divergence.

Figure 3.60 shows the speed lines for cases without particle injection, computed and experimental respectively. It appears that pressure ratio from simulations are lower than those of the experimentally measured points. This discrepancy can be seen on results of other investigations [75]. Usually this discrepancy is noted in these reports but goes unexplained.

Figure 3.61 shows the computed speed lines for the dry case and the two wet compression cases described earlier in this section. The leftmost point of each computed characteristics represents the highest exit pressure at the hub for which the RANS Solver used in the current investigation provided a converged solution. It appears that the computed characteristics corresponding to the two wet compression cases appear to have shifted to the right. As a result, the maximum possible mass flow through the compressor appears to have increased. This shift seems to be proportional to the amount of water injected. Since in both cases full evaporation of all water occurs in the rotor, the magnitude of the evaporative cooling effect brought to the mixture due to evaporation should be proportional to the mass flow of the injected water. As a result, the density of the fluid increases. Thus, the compressor rotor now operates at a higher

corrected speed due to water injection, along with the inter-cooling effect due to the evaporation of the water inside the blade row. The higher the increase of the density of the mixture upstream of the rotor, the higher the corrected speed becomes:

$$n_{corr} = \frac{n}{\sqrt{T_{t1}}} \quad (3.74)$$

Although the rotor inlet total temperature is directly related to the mass flow of the water that evaporates in the rotor, it seems to be difficult to determine the exact correlation between the corrected speed and water injection parameters. One could easily determine the mass-averaged total temperature and pressure at the inlet of the rotor after the simulation, in the post-processing phase.

These results seem to be consistent with the findings of Meacock and White [31], who determined the effect of water injection on speedlines, by coupling mean line (i.e. velocity triangle) calculations with a vaporization code. Some of their findings were the following:

- 1) The characteristics shift to higher mass flow.
- 2) The increment of mass flow is higher than the additional mass flow of water (i.e. the mass flow through the rotor increases due to evaporative cooling.
- 3) The characteristics become steeper and the operating range narrower.

In this investigation, for the case of uniform injection of particles across the span, the mass flow corresponding to rotor stall increased. Thus, the operating range became slightly

Figure 3.60. Computed and Measured Speedlines for NASA Rotor 37.

Figure 3.61. Speedlines for Different Wet Compression Cases and Dry Case.

narrower. One possible explanation for this case can be that now rotor runs at a different corrected speed.

In the case when particles are injected next to the casing, according to Figure 3.61, the opposite is true. In this case, the onset of the numerical stall was delayed to occur at 30000 Pa gauge hub exit pressure, versus 27000 Pa in the case with no particle injection. At this point, the solver indicated reversed flow at the rotor inlet, which ultimately resulted in the divergence of the residuals. The extension of the operating range in this case should be the result of the increase of axial momentum due to evaporative cooling in the tip region, as shown and explained previously with the results seen in Table 3.5.

CHAPTER 4. CONCLUSIONS AND FUTURE WORK

In this study, the behavior of two-phase flow in a transonic compressor rotor (NASA rotor 37) was simulated. Both flow and droplets' governing equations have been formulated and solved in the reference frame of the rotating blades. An Eulerian-Lagrangian approach was used for the continuous and discrete phases, with a two-way interaction model to simulate the mass-, momentum- and energy exchange between the droplets and the fluid.

Water particles were injected at the inlet, with a uniform particle mass flux and a slip velocity of about 20% of the free stream velocity. The particles fully evaporated inside the rotating blade row. The evaporation of water was observed to be the most intense at the leading edge and lambda shock areas, where there is a substantial increase of the slip velocity of the droplets due to their inertia. Results show decreased circumferentially averaged total temperature ratio of the air-vapor mixture across the span, which is a direct result of inter-phase energy coupling. Evaporative cooling increases the density of the gas, resulting in higher mass flow through the rotor, prompting the decrease of the compressor specific work. A water to air mass flow ratio of 1% resulted in a 3% decrease in the compressor specific work at the same pressure ratio. Additionally, the increase of mass flow through a compressor is what leads to a higher net power output of a gas turbine. Two wet cases were studied, one with water particles injected uniformly along the span and the other with the water concentrated near the tip. The effects on the temperature profile at rotor exit were dramatic.

An entropy based approach to calculate the isentropic efficiency of a wet compression process in a transonic compressor rotor was also presented. The viscous dissipation function was calculated everywhere in the domain in the post-processing phase of the simulation and

integrated all the way to the wall, with special treatment in the near-wall regions. High rates of entropy generation occur in this near wall region. The isentropic efficiency of the wet compression is then determined with the entropy generation rate of the rotor already known. For a water to air ratio of 1%, results show increased entropy production across the whole span, resulting in a 5% drop in the wet compressor isentropic efficiency. Analytical integration of wall functions and numerical integration of the viscous dissipation function allows for reasonable results even with relatively coarse grids. The method developed herein can be applied to determine the isentropic efficiency of devices with flow control mechanisms like boundary layer suction or blowing, tip injection from a slot, etc. The entropy-based approach is a thermodynamically sound method to look at losses and loss mechanisms.

A parametric study of the effect of initial particle parameters on the wet compression process was also performed. With increasing particle diameter, the evaporative area of the spray decreases, thus reducing the cooling effect of the phase change on the flow. The decrease of particle injection temperature leads to increased cooling of the compressed air. Increasing the mass flow of the injected particles was shown to saturate some local areas of the rotor, especially in the area between the leading edge shock and passage shock, where the flow is supersonic. This leads to lower total temperature at the rotor exit since total evaporative area of the particles was also increased.

In the last part of the study, speed lines comparing the wet compression cases with the dry compression case were also presented. Results show that numerical stall can be delayed with injection of water near the tip, due to the increase of the axial momentum of the fluid in the endwall region, which is the direct result of phase change.

Future work could include further investigation of compressor operational range enhancement potential of water injection. Another topic would be the investigation of the effects of water ingestion on multi-stage compressor performance. In these compressors, multiple, complex shock structures are present. These could be both computational and experimental works. The investigation of inter-phase interaction on compressor performance at corrected speeds lower than 100% could be another topic to explore.

LIST OF REFERENCES

[1] I. Santa, “Design of Gas Turbine Engines”, Lecture Notes, Technical University of Budapest, Department of Aircraft and Ships, Budapest, Hungary, 1998.

[2] Chaker, M., Meher-Homji, C. B., Thomas Mee III, “Inlet Fogging of Gas Turbine Engines: Experimental and Analytical Investigations on Impaction Pin Fog Nozzle Behavior”, *ASME Journal of Engineering for Gas Turbines and Power*, Vol. 128, pp. 826-839, Oct. 2006.

[3] Daggett, D. L., Hendricks, R. C., “Water Misting & Injection of Commercial Aircraft Engines to Reduce Airport NO_x”, NASA Center for Aerospace Information (CASI) NASA/CR-2004-212957; C/EA-BQ130-Y04-002; E-14397 , 20040301; March 2004, NASA Document ID: 20040035576.

[4] Stalder, J. P., “Gas Turbine Compressor Washing State of the Art: Field Experiences”, *Journal of Engineering for Gas Turbines and Power*, Vol. 123, pp. 363-370, April 2001.

[5] Brun, K., Kurz, R., Simmons, H. R., “Aerodynamic Instability and Life-Limiting Effects of Inlet and Interstage Water Injection Into Gas Turbines”, *Journal of Engineering for Gas Turbines and Power*, Vol. 128, pp. 617-625, July 2006.

[6] Klepper, J., Davis, M., Hale, A. and Hurwitz, W., “A Numerical Investigation of the Effects of Steam Ingestion on Compression System Performance”, ASME Paper GT2004-54190, 2004.

[7] Murthy, S. N. B., 1996, "Dynamic Performance of High Bypass Ratio Turbine Engine with Water Ingestion," NASA document NASA/CR-4703, Washington DC.

[8] Tsuchiya, T., and Murthy, S. N. B., 1982, "WISGSK: A Computer Code for the Prediction of a Multistage Axial Compressor Performance with Water Ingestion." WISGSK: A Computer Code for the Prediction of a Multistage Axial Compressor Performance with Water Ingestion, NASA CR 3624, NASA, Washington.

[9] Meher-Homji, C.B., and Mee, T.R., “Inlet Fogging of Gas Turbine Engines: Part A—Theory, Psychrometrics, and Fog Generation”, ASME Paper 2000-GT-0307, 2000.

[10] Meher-Homji, C.B., and Mee, T.R., “Inlet Fogging of Gas Turbine Engines: Part B—Practical Consideration, Control, and O&M Aspects”, ASME Paper 2000-GT-0308, 2000.

[11] Chaker, M., Meher-Homji, C.B., Mee T.R. III, “Inlet Fogging of Gas Turbine Engines: Part C—Fog Behavior in Inlet Ducts, CFD Analysis and Wind Tunnel Experiments”, ASME Paper GT-2002-30564, 2002.

[12] Chaker, M., Meher-Homji, C.B., Mee T.R. III, "Inlet Fogging of Gas Turbine Engines: Part A-Fog Droplet Thermodynamics, Heat Transfer and Practical Considerations", ASME Paper GT-2002-30562, 2002.

[13] Chaker, M., Meher-Homji, C.B., Mee T.R. III, "Inlet Fogging of Gas Turbine Engines: Part B-Fog Droplet Sizing Analysis, Nozzle Types, Measurement and Testing", ASME Paper GT-2002-30563, 2002.

[14] Loebig, J., B. Vittal and M. Booher, "Numerical Simulation of Water/Methanol Evaporation in an Axial Flow Gas Turbine Compressor", AIAA Paper 1998-3559, 1998.

[15] Ulrichs, E., F. Joos, "Experimental Investigations of the Water Droplets in Compressor Cascades", ASME Paper GT2006-90411, 2006.

[16] Mund, F. C. and P. Pilidis, "Effects of Spray Parameters and Operating Conditions on an Industrial Gas Turbine Washing System", ASME Paper GT2004-53551, 2004.

[17] Mathioudakis, K., "Analysis of the Effects of Water Injection on the Performance of a Gas Turbine.", *Journal of Engineering for Gas Turbines and Power*, Vol. 124, No. 3., pp. 489-495, 2002.

[18] Zheng, Q. and M. Li, "Wet Compression System Stability Analysis : Part II - Simulations and Bifurcation Analysis.", ASME Paper GT2004-54018, 2004.

[19] Zheng, Q. and M. Li , "Wet Compression System Stability Analysis : Part II - Simulations and Bifurcation Analysis. ", ASME Paper GT2004-54020, 2004.

[20] Jolly, S., "Wet Compression-A Powerful Means of Enhancing Combustion Turbine Capacity", Presented at Power-Gen. International, Orlando, Florida, 2002.

[21] Jolly, S., "Performance Enhancement of GT 24 with Wet Compression", Presented at Power-Gen. International, Las Vegas, NV, 2003.

[22] Seton, W. R., Sexton, M. R., "The Effects of Wet Compression on Gas Turbine Engine Operating Performance", ASME Paper GT-2003-38045, 2003.

[23] Wang, T., X. Li and Pinninti, V., "Simulation of Mist Transport for Gas Turbine Inlet Cooling", ASME Paper IMEGE-2004-60133, 2004.

[24] Hartel, C. and P. Pfeiffer, "Model Analysis of High-Fogging Effects on the Work of Compression", ASME Paper GT-2003-38117, 2003.

[25] Zainol, M., P. Pilidis and J. Teixeira, "CFD Aerodynamic Investigation of Air-Water Trajectories on Rotor-Stator Blade of an Axial Compressor for Online Washing", ASME Paper GT2006-90745, 2006.

[26] Das, K., Hamed, A. & Basu, D., "Droplet Trajectories and Collection on Fan Rotor at Off-Design Conditions" ASME paper GT 2006-91214, 2006.

[27] Hamed, A., Das, K., and Basu, D., "Numerical simulations of ice droplet trajectories and collection efficiency on aero-engine rotating machinery," AIAA Paper 2005-1248, 2005.

[28] Das, K., Hamed, A. & Basu, D., "Ice Shape Prediction For Turbofan Rotating Blades," AIAA Paper 2006-0209, 2006.

[29] Zheng, Q., Y. Sun, S. Li and Y. Wang "Thermodynamic analyses of wet compression process in the compressor of gas turbine", ASME Paper GT2002-30590, 2002.

[30] Shao, Y. and Zheng, Q., "The entropy and energy analyses of wet compression gas turbine", ASME Paper GT2005-68649, 2005.

[31] White, A. J., Meacock, A. J., "An Evaluation of the Effects of Water Injection on Compressor Performance," ASME Paper 2003-GT-38237, 2003.

[32] Schuler, B. J., J. L. Kerrebrock and A. Merchant, "Experimental Investigation of a High Pressure Ratio Aspirated Fan Stage", ASME Paper GT-2004-53679, 2004.

[33] Suder, K. L., Hathaway, M. D., Thorp, S. A., Strazisar, A. J., and Bright, M.M., "Compressor Stability Enhancement Using Discrete Tip Injection," *ASME Journal of Turbomachinery*, Vol. 123, No. 1, pp. 14-23, 2001.

[34] Strazisar, A. J., Bright, M. M., S. Thorp, Culley, D. E., Suder, K. E., “Compressor Stall Control Through Endwall Recirculation”, ASME paper GT-2004-54295, 2004.

[35] Weigl, H. J., Paduano, J. D., Frechette, L. G., Epstein, A. H., Greitzer, E. M., Bright, M.M., and Strazisar, A.J., "Active Stabilization of Rotating Stall and Surge in a Transonic Single Stage Axial Compressor," *ASME Journal of Turbomachinery*, Vol. 120, No. 4, pp. 625-636, 1998.

[36] Spakovszky, Z.S., Weigl, H.J., Paduano, J.D., van Schalkwyk, C.M., Suder, K.L., and Bright, M.M., “Rotating Stall Control in a High-Speed Stage with Inlet Distortion: Part I – Radial Distortion,” *ASME Journal of Turbomachinery*, Vol. 121, pp. 510-516, 1999.

[37] I. Wilke and H.-P. Kau, “A Numerical Investigation of the Flow Mechanisms in a HPC Front Stage with Axial Slots”, ASME Paper GT2003-38481, 2003.

[38] G.D.J. Smith, N. A. Cumpsty, “Flow Phenomena in Compressor Casing Treatment”, *ASME Journal of Eng. Gas Turbines and Power*, Vol. 106, No. 3, pp. 532-541, 1984.

[39] Moore, R. D.; Reid, L., “Performance of single-stage axial-flow transonic compressor with rotor and stator aspect ratios of 1.19 and 1.26 respectively, and with design pressure ratio of 2.05” ,1980, NASA TP-1659, NASA Document ID: 19800012840.

[40] Moore, R. D.; Reid, L., “Design and Overall Performance of Four Highly-Loaded, High-Speed Inlet Stages for an Advanced, High-Pressure-Ratio Core Compressor”, 1978, NASA TP-1337.

[41] Crowe, Clayton, T., M. Sommerfeld, and Yutaka Tsuji, “Multiphase Flows with Droplets and Particles”, CRC Press, 1998.

[42] Fluent Users Manual Version 6.3., www.fluentusers.com

[43] Crowe, C. T. , T.R. Troutt and J.N. Chung, "A Turbulent Flow Without Particle Mixing", *Journal of Aerosol Science and Technology*, Vol. 22, pp. 135-138, 1995.

[44] Sirignano, W. A., “Fluid Dynamics and Transport of Droplets and Sprays”, Cambridge University Press, Cambridge, UK, 1999, pp. 12-13.

[45] Goossens, H.W.J., J. W. Cleijne, H. J. Smolders, and M. E. H. van Dongen, 1988, “Shock wave induced evaporation of water droplets in a gas-droplet mixture”, *Experiments in Fluids*, Vol. 6, pp. 561-568, Springer-Verlag, 1988.

[46] Crowe, C. T., Sharma, M. P. and Stock, D. E., “The Particle Source in Cell Method for Gas Droplet Flow”, *Journal of Fluid Engineering*, Vol. 99, page 325, 1977.

[47] Ranz, W.E. and Marshall, W.R., 1952, “Evaporation from Drops”, *Chemical Engineering Progress*, Vol. 48, pp. 141-46 and 173-80.

[48] Crowe, C. T., 1976, “Conservation Equations for Vapor-Droplet Flows including Boundary Droplet Effects”, Lawrence Livermore Laboratory, Livermore, California, UCRL-52184.

[49] Lukovic, B., “Modeling Unsteadiness in Steady Simulations with Neural Networks”, PhD Dissertation, 2002, University of Cincinnati, Cincinnati, OH

[50] Howell, A. R., “Fluid Dynamics of Axial Compressors,” *Proc. Inst. Mech. Eng.*, 153, pp. 441–452, 1945.

[51] Denton J.D., “Loss Mechanisms in Turbomachines”, *ASME Journal of Turbomachinery*, Vol. 115(4), pp.621-656, 1993.

[52] Shah, P. N. and C.S. Tan “Effect of blade passage surface heat extraction on axial compressor performance”, ASME Paper GT-2005-68815, 2005.

[53] Jennions, I. K., and J. J. Adamczyk, “Evaluation of the Interaction Losses in a Transonic Turbine HP Rotor/ LP Vane Configuration”, ASME Paper 95-GT-299, 1995.

- [54] Bejan, Adrian, Entropy Generation Minimization, CRC Press, Boca Raton, FL, 1996, pp. 22-23.
- [55] Hill, H.E., W. F. Ng, and P. P. Vlachos, S. Guillot, D. Car, “2D Parametric Study using CFD of a Circulation Control Inlet Guide Vane”, ASME Paper GT-2007-28058, 2007.
- [56] Kirtley, K.R., P. Graziosi, P. Wood, B. Beacher, H. W. Shin “Design and Test of an Ultra-Low Solidity Flow-Controlled Compressor Stator”, ASME Paper GT-2004-53012, 2004.
- [57] Kock, Fabian and Heinz Herwig, “Local Entropy Production in Turbulent Shear Flows: a High Reynolds Number Model with Wall Functions”, *International Journal of Heat and Mass Transfer*, Vol. 47, pp. 2205-2215, 2004.
- [58] Kock, F and H. Herwig , “Entropy Production for Turbulent Shear Flows and their Implementation in CFD Codes” CHT Paper 04-112, 1994.
- [59] Herwig, H and F. Kock “Direct and Indirect Methods of Calculating Entropy Generation Rates in Turbulent Convective Heat Transfer Problems”, Springer-Verlag *Journal of Heat Mass Transfer* , Vol. 43, pp. 207–215, 2007.
- [60] Moore, J., and Moore, J. G., “Entropy Production Rates from Viscous Flow Calculations, Part I, A Turbulent Boundary Layer Flow,” ASME Paper 83-GT-70, 1983.

- [61] Bejan, A., Convection Heat Transfer, 2nd edition, Wiley, New York, 1995.
- [62] Kawamura, H., H. Abe, Y. Matsuo, “DNS of Turbulent Heat Transfer in Channel with Respect to Reynolds and Prandtl Number Effects”, *Int. Journal of Heat Fluid Flow*, Vol. 20, pp. 196-207, 1999.
- [63] Schlichting, H. “Boundary Layer Theory”, 7th Edition, McGraw-Hill, New York, 1978.
- [64] Shabbir, A. and M. Turner, “A Wall Function for Calculating the Skin Friction With Surface Roughness”, ASME Paper GT-2004-53908, 2004.
- [65] Bejan, Adrian, “Entropy generation minimization”, CRC Press, 1996, pp. 52-55.
- [66] Young, J.B., “The Fundamental Relations of Gas-Droplet Multiphase Flows”, *Int. Journal of Multiphase Flow*, Vol. 21, Nr. 2, pp. 175-191, 1995.
- [67] Tabakoff, W., S. Kaushik, A. N. Lakshminarasimha, “Performance Improvement of an Eroded Axial Flow Compressor Using Water Injection”, AIAA Paper 1990-2016, 1990.
- [68] Meacock, A.J. and A. J. White, “The Effect of Water Injection on Multi Spool Gas Turbine Behavior”, *Journal of Engineering for Gas Turbines and Power*, Vol. 128, pp. 97-102, January 2006.

[69] Roumeliotis, I. and K. Mathioudakis, “Water Injection Effects on Compressor Stage Operation”, ASME Paper GT-2006-90427, 2006.

[70] Adamczyk, J. J., Celestina, M.L. and Greitzer, E. M., “The Role of Tip Clearance in High-Speed Fan Stall”, *ASME Journal of Turbomachinery*, Vol. 115, pp. 28-39, 1993.

[71] Kirtley, K. R., Beach, T. A. and Adamczyk, J. J., “Numerical Analysis of Secondary Flow in a Two-Stage Turbine”, AIAA Paper 90-2356, 1990.

[72] Furukawa, M., Inoue, M., Saiki, K., Yamada, K., “The Role of Tip Leakage Vortex Breakdown on Compressor Rotor Aerodynamics”, *ASME Journal of Turbomachinery*, Vol. 121, No. 3, pp. 469-480, 1999.

[73] Crook, A. J., Greitzer, E. M., Tan, C. S., Adamczyk, J. J., “Numerical Simulation of Compressor Endwall and Casing Treatment Flow Phenomena”, *ASME Journal of Turbomachinery*, Vol. 115, pp. 501-512, 1993.

[74] Jennions, I. K., Turner, M. G., “Three-Dimensional Navier-Stokes Computations of Transonic Fan Flow using an Explicit Flow Solver and an implicit k- ϵ solver”, *ASME Journal of Turbomachinery*, Vol. 115, p.263, April 1993.

[75] Chima, R. V. , “Calculation of Tip Clearance Effect in a Transonic Compressor Rotor”, *ASME Journal of Turbomachinery*, Vol.120, pp. 131-140, 1998.

[76] Shabbir, A. and J. J. Adamczyk, “Flow Mechanism for Stall Margin Improvement due to Circumferential Casing Grooves on Axial Compressors”, ASME Paper GT2004-53903, 2004.

APPENDIX A: Multi-Phase Model Validation

In order to validate the Eulerian-Lagrangian multi-phase solver used in the current investigation, several numerical test cases were performed.

First, NASA Transonic Compressor Rotor 37 was simulated at design conditions, without water injection. The grid consisted of $132 \times 51 \times 41$ grid points with two cells in the tip gap. Boundary conditions are those described in Section 2.2.3. Blade-to-blade relative Mach number plots at several span locations were validated against measurement data available. In addition, circumferentially averaged quantities were calculated along the span and with experimental results.

Figures A.1 and A.2 compare the radial profiles of design flow variables to the circumferential average of the rotor exit computational results. According to these plots, computed isentropic efficiency seems to be in close agreement with design data. On the other hand, the computed total pressure ratio is slightly higher than the design value. No attempt was made to change the specified static pressure at the hub in order to get closer to the design pressure ratio. The circumferentially averaged temperature ratio was calculated the following way:

$$\frac{T_{t2}}{T_{t1}} = \frac{\sum_{i=1}^{41} \frac{T_{t2}}{T_{t1}} \cdot \rho \cdot u \cdot A}{\sum_{i=1}^{41} \rho \cdot u \cdot A} \quad (\text{A.1})$$

Figure A.1. Circumferentially Averaged Total Pressure Ratio for NASA Rotor 37.

Figure A.2. Circumferentially Averaged Isentropic Efficiency for NASA Rotor 37.

The circumferential average of the total pressure ratio was calculated using the following formula, which is sometimes referred to as an enthalpy-weighted average:

$$\frac{p_{t2}}{p_{t1}} = \left(\frac{\sum_{i=1}^{41} \rho \cdot u \cdot A \cdot \left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma}}}{\sum_{i=1}^{41} \rho \cdot u \cdot A} \right)^{\frac{\gamma}{\gamma-1}} \quad (\text{A.2})$$

This comparison demonstrates that although the grid is coarse, it is acceptable for these transonic rotor calculations, i.e. when using this grid for simulation, results would be reasonable.

A.1. Grid Independence Study

A grid independence study was performed using the same boundary and initial conditions, in which three different grid resolutions were used.

The coarse grid consisted of 132 x 41 x 31 grid points in the streamwise, hub-to-tip and the blade-to-blade directions, with two cells in the tip gap. The medium grid consisted of 132 x 51 x 41 grid points with two cells in the tip gap. Finally, the fine grid consisted of 132 x 70 x 100 grid points with seven cells in the tip gap.

Computed circumferentially averaged quantities for these three cases along the span are compared on Figures A.3 and A.4. The differences between these plots can be considered relatively small in magnitude, considering the large influence the evaporation has, which is the focus of this research.

Figure A.3. Circumferentially Averaged Total Pressure Ratio for NASA Rotor 37, Convergence Study.

Figure A.4. Circumferentially Averaged Total Temperature Ratio for NASA Rotor 37, Convergence Study.

A.2. Behavior of the Two-Phase Model

An attempt was made to understand the behavior of the two-way coupled Eulerian-Lagrangian multiphase model used for the current investigation. For this purpose, a simple, quasi one-dimensional, inviscid, compressible two-dimensional test case was validated.

Figure A.5 shows the test geometry, a simple pipe, along with its dimensions and boundary. Standard atmospheric total pressure and temperature were specified at the entrance of the pipe, while a static pressure lower than atmospheric was specified at the exit. Thus, the flow across the pipe is driven solely by pressure gradient, resulting in a constant flow speed of 34 m/s across the pipe when no particles are present. Since adiabatic boundary conditions were specified at the walls, the total specific enthalpy of the flow remained constant along the pipe.

In order to see whether basic conservation laws hold for a control volume that includes the pipe, ten particle streams were injected uniformly at the inlet of the pipe, parallel to the longitudinal axis. The initial parameters of the particles were carefully selected based on numerical experiments performed earlier with the code. Particle temperature was 300 K at the pipe inlet in order to maximize water vapor concentration at the surface of the particle, thus maximizing the driving potential of the mass and energy exchange between the phases.

It is possible to analyze the system as a closed system from a thermodynamics perspective. Figure A.6 illustrates the control volume enclosing the pipe. In order to see whether basic conservation laws are held for the pipe, one can rewrite the thermal energy equation for the mixture, assuming steady flow, low particle volume fraction and low relative Mach number:

$$\bar{\nabla} \cdot (\rho \bar{v} h_t) = -\frac{1}{V} \sum_j \dot{m}_j h_{s,j} + \bar{\nabla} \cdot (\bar{\tau} \cdot \bar{v}) + \bar{\nabla} (k \bar{\nabla} T) + \rho \cdot \bar{v} \cdot \bar{g} - \frac{1}{V} \sum_j \dot{q}_j \quad (\text{A.3})$$

Since the flow in the pipe is inviscid, the second term on the right hand side is equal to zero. The third term is small enough to be neglected (due to adiabatic boundaries and the fact that there is no heat source on the domain). Assuming that there is no gravity acting on particles the fourth term is zero too. Thus, the thermal energy equation for the mixture in the pipe becomes:

$$\bar{\nabla} \cdot (\rho \bar{v} h_t) = -\frac{1}{V} \sum_j \dot{m}_j h_{s,j} - \frac{1}{V} \sum_j \dot{q}_j \quad (\text{A.4})$$

By integrating Equation (A.4) with respect to the volume enclosing by the pipe, and applying Green's theorem on the left hand side, one gets the following enthalpy balance for the mixture:

$$\dot{m}_2 h_{t,2} - \dot{m}_1 h_{t,1} = -\int_V \left[\frac{1}{V} \sum_j \dot{m}_j h_{s,j} + \frac{1}{V} \sum_j \dot{q}_j \right] dV \quad (\text{A.5})$$

Evaluating this expression in the post-processing phase by integrating the right hand side numerically shows that Equation (A.4) holds for the pipe flow case, which is one indication that the Eulerian-Lagrangian solver gives physically correct results.

Since the expression on the right hand side is the heat released by the particles, if a control volume analysis is conducted on the rotor that includes particles, the result is:

$$H_2 = H_1 + \dot{m}_p c_D T_p \quad (\text{A.6})$$

This is exactly the result obtained when the control volume representing the pipe (Figure A.6) is examined. From mass conservation perspective, the following should be true:

$$\dot{m}_1 + \dot{m}_D = \dot{m}_2 \quad (\text{A.7})$$

As it can be seen from Figure A.6, the equation of mass conservation also holds true for the control volume representing the pipe.

The validation processes described in the current section gives confidence that results obtained during the simulation of particle injection into the rotor would be reasonable.

T_{t, inlet} =300 K

P_{t, inlet}=1 atm

Pipe length=10 m

Pipe diameter=0.3 m

Figure A.5. Illustration of the Pipe Test Case.

Figure A.6. Control Volume Analysis of the Pipe.

APPENDIX B: Effect of Initial Particle Temperature on Inter-Phase Energy Exchange

By rearranging the energy equation for an individual droplet, one can arrive at the following expression:

$$\frac{d(m_p c_p T_p)}{dt} = h A_p (T - T_p) + \frac{dm_p}{dt} h_s \quad (\text{B.1})$$

Assume that the particle exchanges mass, momentum and energy with the mixture. Let the initial and final states of the particle be denoted by subscripts **1** and **2** respectively. At state **2**, the diameter of the particle is zero, i.e. it has fully evaporated. After re-arranging Equation (B.1) and integrating both sides with respect to time one gets:

$$m_p c_p T_p \Big|_1^2 = q + m_p \Big|_1^2 h_s \quad (\text{B.2})$$

If a particle evaporates fully in the rotor, the total sensible heat exchange between the phases becomes:

$$q = (m_p c_p T_p)_1 - (m_p)_1 h_s \quad (\text{B.3})$$

In Fluent, the sensible heat exchange between the phases for a particle is calculated as:

$$S_{ener} = m_{p,in} c_p T_{p,in} - m_{p,out} c_p T_{p,out} - h_{fg} (m_{p,in} - m_{p,out}) \quad (\text{B.4})$$

Equation (B.4) shows that the sensible heat extracted from the mixture decreases with increasing droplet initial temperature.

APPENDIX C: Fluent Solution Settings

The following are the solution settings within FLUENT for all the wet compression simulations using NASA Compressor Rotor 37:

1. DEFINE->MODEL->SOLVER

Coupled, implicit, steady

2. DEFINE->MODEL->ENERGY

Energy->enable

3. DEFINE->MODELS->VISCOUS

k- ϵ (two equation)->enable with defaults

4. DEFINE->MATERIALS

Material type->mixture

Density->ideal gas

Cp->mixing law

Mixture species->edit names

Make sure in the selected species column there are h20 and air selected in this order

4. DEFINE->MODELS->SPECIES->TRANSPORT & REACTION

Enable species transport

5. DEFINE->INJECTIONS->CREATE

Select droplet option

Material: Water liquid

Evaporating species: h20

Select injection type (surface) and specify initial conditions for the droplets as well as surface of injection (rotor inlet)

6. DEFINE->MODELS->DDISCRETE PHASE

Enable interaction with continuous phase

DPM iterations to continuous phase iterations: set to 2

7. SOLVE->CONTROLS->SOLUTION

Set accuracy to second order for flow

8. SOLVE->CONTROLS->INITIALIZE

Absolute, all zones, accept default values, except type in 0.01 for h20

9. SOLVE->CONTROLS->INITIALIZE

Reset DPM sources

10. Solve->Iterate

APPENDIX D: Grid File Format Conversion

The code converts a three-dimensional unstructured plot3d format grid file into 3D structured ASC format, which then can be read into Gambit.

```
#include <stdio.h>
#include <stdlib.h>
#include <iostream.h>

int main () {
    FILE *pInFile = fopen ( "bladel1.bin" , "rb" );
    FILE *pOutFile = fopen ( "uj37.g" , "w" );
    long lSize;
    float buffer;
    int head[3];

    // obtain file size.
    fseek (pInFile , 0 , SEEK_END);
    lSize = ftell (pInFile);
    rewind (pInFile);

    //read in bytes from file

    fread( head, 4, 3, pInFile );
    int ii;
    for ( ii=0; ii<3; ++ii )
    {
        fprintf( pOutFile, "%d ", head[ii] );
        --lSize;
    }
    //write out grid in a structured format
    fprintf( pOutFile, "\n" );

    int LineNum = lSize/4;
    for ( ii=0; ii<LineNum; ++ii )
    {
        for ( int jj=0; jj<4; ++jj )
        {
            fread( &buffer, 4, 1, pInFile );

            fprintf( pOutFile, "%e ", buffer );
        }
        fprintf( pOutFile, "\n" );
    }

    fclose (pInFile);
    fclose( pOutFile );
    return 0;}
```

APPENDIX E: Circumferential Averaging

```
#include <iostream.h>
#include <stdio.h>
#include <stdlib.h>
#include <math.h>

int main(int argc, char* argv[])
{

//define pointers corresponding flow variables and allocate memories

char *input_filename1=(char *) malloc(50000);
char *input_filename2=(char *) malloc(50000);
char *output_filename=(char *) malloc(50000);
char *str=(char *) malloc(50000);
char vesszo=' ';
int *number_of_features=(int *) malloc(40000*sizeof(int));

double *cell_number=(double *) malloc(2003*sizeof(double));
double *X=(double *) malloc(2003*sizeof(double));
double *Y=(double *) malloc(2003*sizeof(double));
double *Z=(double *) malloc(2003*sizeof(double));
double *R=(double *) malloc(2003*sizeof(double));
double *P_t=(double *) malloc(2003*sizeof(double));
double *RO=(double *) malloc(2003*sizeof(double));
double *U=(double *) malloc(2003*sizeof(double));
double *T_t=(double *) malloc(2003*sizeof(double));
double *A_cell=(double *) malloc(2003*sizeof(double));
double *P_t_ratio_szer_massflow=(double *) malloc(2003*sizeof(double));
double *T_t_ratio_szer_massflow=(double *) malloc(2003*sizeof(double));

double *P_t_circum_avg=(double *) malloc(49*sizeof(double));
double *T_t_circum_avg=(double *) malloc(49*sizeof(double));
double *sum_massflow=(double *) malloc(49*sizeof(double));

double *massflow=(double *) malloc(2003*sizeof(double));

struct memory{

int feature_count;
float time[100];

}*s;

s=(struct memory*) malloc(4000*sizeof(struct memory));
```

```

cout<<"enter the name of the text file containing data"<<endl;
cin>>input_filename1;

    ofstream output_file(output_filename);
    fstream file_op1(input_filename1,ios::in);

float g;
double d;
int counter=0;
int global_counter=0;

//reading in data from file containing results at rotor exit

    while(!file_op1.eof())
    {
        file_op1.getline(str,70,',');

        sscanf(str,"%g",&g);

        d=atof(str);

        counter++;

        if (counter==1)    {            global_counter++;}

        switch(counter)    {

            case 1: cell_number[global_counter]=d;

            break;

            case 2: X[global_counter]=d;

            break;

            case 3: Y[global_counter]=d;

            break;

            case 4: Z[global_counter]=d;

            break;

            case 5: P_t[global_counter]=d;

```

```

        break;

        case 6: RO[global_counter]=d;
        break;

        case 7: U[global_counter]=d;
        break;

        case 8: T_t[global_counter]=d;
        break;

        case 9: A_cell[global_counter]=d;
        break;

    }

    if (counter==9)    {counter=0;}

}

//calculating circumferebtially averaged quantities along span

    for (int t=0;t<2001;t++)    {
P_t_ratio_szer_massflow[t]=RO[t]*U[t]*A_cell[t]*(P_t[t])/101325;
T_t_ratio_szer_massflow[t]=RO[t]*U[t]*A_cell[t]*(T_t[t]/288.0);
massflow[t]=RO[t]*U[t]*A_cell[t];
R[t]=sqrt(Z[t]*Z[t]+Y[t]*Y[t]);
    }

double
R_values[49]={0.1939,0.1940,0.1941,0.1942,0.1943,0.1944,0.1946,0.1948,0.1950,
0.1953,

```



```

0.1956,0.1960,0.1965,0.1970,0.1977,0.1985,0.1995,0.2006,0.2020,0.2037,0.2057,
0.2080,0.2109,
0.2143,0.2181,0.2215,0.2243,0.2244,0.2267,0.2287,0.2304,0.2318,0.2329,0.2339,
0.2347,0.2354,
0.2359,0.2364,0.2368,0.2371,0.2374,0.2376,0.2378,0.2380,0.2381,0.2382,0.2383,
0.2384,0.2385};

```

```

for (t=1;t<=49;t++)      {

```

```

    P_t_circum_avg[t]=0;
    T_t_circum_avg[t]=0;
    sum_massflow[t]=0;

```

```

}

```

```

int szamlalo;
for (t=0;t<49;t++)      {

double P_t_circum_elozo=0;
double T_t_circum_elozo=0;
double sum_massflow_elozo=0;

```

```

szamlalo=0;

```

```

for (int k=1;k<2001;k++)      {

    if (fabs(R_values[t]-R[k])<0.0001) {

        sum_massflow[t]=sum_massflow_elozo+massflow[k];
        P_t_circum_avg[t]=P_t_circum_elozo+P_t_ratio_szer_massflow[k];
        T_t_circum_avg[t]=T_t_circum_elozo+T_t_ratio_szer_massflow[k];
        P_t_circum_elozo=P_t_circum_avg[t];
        T_t_circum_elozo=T_t_circum_avg[t];
        sum_massflow_elozo=sum_massflow[t];

    }

```

```

    }

```

```

}

```

```

for (t=0;t<49;t++)      {

```

```

    cout<<t<<"    "<<P_t_circum_avg[t]/sum_massflow[t]<<"
    "<<T_t_circum_avg[t]/sum_massflow[t]<<endl;

```

```

    }

```

```

    file_op1.close();

```

```

return 0;}

```