Electronic Theses and Dissertations, 2004-2019

2004

Effect Of Coriolis And Centrifugal Forces On Turbulence And Transport At High Rotation And Buoyancy Numbers

Ahmad Khalaf Sleiti

University of Central Florida

Part of the Engineering Commons

Find similar works at: https://stars.library.ucf.edu/etd

University of Central Florida Libraries http://library.ucf.edu

This Doctoral Dissertation (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations, 2004-2019 by an authorized administrator of STARS. For more information, please contact STARS@ucf.edu.

STARS Citation

https://stars.library.ucf.edu/etd/118
EFFECT OF CORIOLIS AND CENTRIFUGAL FORCES ON TURBULENCE AND TRANSPORT AT HIGH ROTATION AND BUOYANCY NUMBERS

by

AHMAD K. SLEITI
M.Sc. Rostov Engineering Institute, 1991

A dissertation submitted in partial fulfillment of the requirements
for the degree of Doctor of Philosophy
in the Department of Mechanical, Materials and Aerospace Engineering
in the College of Engineering and Computer Science
at the University of Central Florida
Orlando, Florida

Spring Term
2004
ABSTRACT

This study attempts to understand one of the most fundamental and challenging problems in fluid flow and heat transfer for rotating machines. The study focuses on gas turbines and electric generators for high temperature and high energy density applications, respectively, both of which employ rotating cooling channels so that materials do not fail under high temperature and high stress environment.

Prediction of fluid flow and heat transfer inside internal cooling channels that rotate at high rotation number and high density ratio similar to those that are existing in turbine blades and generator rotors is the main focus of this study. Both smooth-wall and rib-roughened channels are considered here. Rotation, buoyancy, bends, ribs and boundary conditions affect the flow inside these channels. Ribs are introduced inside internal cooling channel in order to enhance the heat transfer rate. The use of ribs causes rapid increase in the supply pressure, which is already limited in a turbine or a generator and requires high cost for manufacturing. Hence careful optimization is needed to justify the use of ribs. Increasing rotation number (Ro) is another approach to increase heat transfer rate to values that are comparable to those achieved by introduction of ribs. One objective of this research is to study and compare these two approaches in order to decide the optimum range of application and a possible replacement of the high-cost and complex ribs by increasing Ro.
A fully computational approach is employed in this study. On the basis of comparison of two-equation (k-ε and k-ω) and RSM turbulence models against limited available experimental data, it is concluded that the two-equation turbulence models cannot predict the anisotropic turbulent flow field and heat transfer correctly, while RSM showed improved prediction. For the near wall region, two approaches with standard wall functions and enhanced near wall treatment were investigated. The enhanced near wall approach showed superior results to the standard wall functions approach. Thus RSM with enhanced near wall treatment is validated against available experimental data (which are primarily at low rotation and buoyancy numbers). The model was then used for cases with high Ro (as much as 1.29) and high-density ratios (DR) (up to 0.4). Particular attention is given to how turbulence intensity, Reynolds stresses and transport are affected by Coriolis and buoyancy/centrifugal forces caused by high levels of Ro and DR. Variations of flow total pressure along the rotating channel are also predicted. The results obtained are explained in view of physical interpretation of Coriolis and centrifugal forces.

Investigation of channels with smooth and with rib-roughened walls that are rotating about an orthogonal axis showed that increasing Ro always enhances turbulence and the heat transfer rate, while at high Ro, increasing DR although causes higher turbulence activity but does not necessarily increase Nu and in some locations even decreases Nu. The increasing thermal boundary layer thickness near walls is the possible reason for this behavior of Nu. The heat transfer enhancement for smooth-wall cases correlates linearly with Ro (with other parameters are kept constant) and hence it is possible to derive linear correlation for the increase in Nu as a function of Ro. Investigation of channels with rib-roughened walls that rotate about orthogonal axis showed that 4-side-average Nur correlates with Ro linearly, where a linear correlation for
Nur/Nus as a function of Ro is derived. It is also observed that the heat transfer rate on smooth-wall channel can be enhanced rapidly by increasing Ro to values that are comparable to the enhancement due to the introduction of ribs inside internal cooling channels. This observation suggests that ribs may be unnecessary in high-speed machines, and has tremendous implications for possible cost savings in these machines.

In square channels that rotate about parallel axis, the heat transfer rate increases with Ro on three surfaces of the square channel and decreases on the inner surface (that is the one closest to the axis of rotation). However, the four-side average Nu increases with Ro. Increasing wall heat flux at high Ro does not necessarily increase Nu on walls although higher turbulence activity is observed.

This study examines the rich interplay of physics under the simultaneous actions of Coriolis and centrifugal/buoyancy forces in one of the most challenging internal flow configurations. Several important conclusions are reached from this computational study that may have far-reaching implications on how turbine blades and generator rotors are currently designed. Since the computation study is not validated for high Ro cases, these important results call for experimental investigation.
This dissertation is first and foremost dedicated to my wife and our kids, Rabi and Dalia. I would also like to dedicate this dissertation to the people whose influence and support knowingly or unknowingly, molded my personality and contributed to my success.
ACKNOWLEDGMENTS

I would like to express my heartfelt gratitude to my advisor Dr. J. Kapat for his support, advice and guidance throughout my doctoral study. Working with him was a rewarding experience.

I also would like to thank my committee members, Dr. Kassab, Dr. Ilegbusi, Dr. Kumar, Dr. Scott and Dr. Shivamoggi for their valuable comments, discussions, directions and time reviewing my dissertation. Thanks are due to all my professors at University of Central Florida and to everyone who had contributed to my career.

Special thanks to Mr. Jay Vaidya, owner of Electrodynamics Associates INC, for his support and guidance. Working for him on the Advanced Cooling Technology for High Density Low Cycle Generator project and on other mechanical and thermal projects was a rewarding experience and a great contribution to my career.
TABLE OF CONTENTS

LIST OF FIGURES ........................................................................................................... xiv
LIST OF TABLES ................................................................................................................ xix
LIST OF ABBREVIATIONS ............................................................................................ xx
LIST OF VARIABLES ..................................................................................................... xxii

CHAPTER ONE: INTRODUCTION ...................................................................................... 1
  1.1 Background ............................................................................................................. 1
  1.2 Internal Cooling of Turbine Blades ....................................................................... 2
  1.3 Internal Cooling of Rotors of Electrical Machines ............................................... 7
  1.4 Problem Statement ............................................................................................... 9
  1.5 Methodology ......................................................................................................... 11
  1.6 Scope .................................................................................................................... 12
  1.7 Uniqueness of the Research and its Contribution .................................................. 12

CHAPTER TWO: GOVERNING EQUATIONS, TURBULENCE AND HEAT TRANSFER MODELS .................................................................................................................................. 14
  2.1 Governing Equations ............................................................................................ 14
    2.1.1 Continuity Equation ....................................................................................... 14
    2.1.2 Momentum Equation .................................................................................... 14
    2.1.3 Energy Equation .......................................................................................... 15
    2.1.4 Acceleration Terms due to Rotation ............................................................. 15
2. 2 Turbulence Models ........................................................................................................... 16
   2.2 1 Direct Numerical Simulations.......................................................................................... 16
   2.2 2 Large Eddy Simulation (LES) Model........................................................................... 17
   2.2 3 Reynolds Average Navier Stokes Models, (RANS).................................................. 18
       2.2.3 1 Eddy-Viscosity Models (EVMs)........................................................................... 18
           2.2.3.1. 1 Boussinesq Hypothesis ............................................................................. 18
           2.2.3.1. 2 Two-Equation EVMs .................................................................................. 20
       2.2.3 2 Reynolds Stress Models......................................................................................... 24
   2. 3 Turbulent Heat Transfer Models.................................................................................... 27

CHAPTER THREE: MODELING CONSIDERATIONS OF FLOW PHENOMENA IN
ROTATING CHANNELS............................................................................................................. 29

   3. 1 Cooling Channels Walls ................................................................................................. 29
       3.1 1 Modeling Wall Roughness Effects .......................................................................... 31
       3.1 2 Near Wall Treatment ............................................................................................... 32
   3. 2 Effect of Ribs .................................................................................................................. 36
   3. 3 Secondary Flows.............................................................................................................. 40
   3. 4 U-Bends – Effect of curvature ......................................................................................... 42
   3. 5 Rotational Effects........................................................................................................... 44

CHAPTER FOUR: LITERATURE REVIEW ............................................................................... 46

   4. 1 Work with Reference to Orthogonal Mode Rotation.................................................... 46
       4.1 1 Channels with Smooth Walls..................................................................................... 47
           4.1.1 1 Effect of Rotation Number in Smooth Wall Channels - Experimental Work
           ................................................................................................................................. 47
4.1.1 2 Effect of Rotation Number in Smooth Wall Channels - Numerical Simulations ................................................................. 48
4.1.1 3 Effect of Density Ratio in Smooth Wall Channels - Experimental Work 49
4.1.1 4 Combined Effects of Rotation Number and Density Ratio- Experimental Work ............................................................................................................................ 50
4.1.1 5 Combined Effects of Rotation Number and Density Ratio- Numerical Simulations: ................................................................. 51

4.1.2 Rib Roughened -Wall Coolant Passage ........................................................ 51
4.1.2.1 Channels with 90 Ribs- Experimental Work ........................................ 52
4.1.2.2 Channels with 90 Ribs-Numerical Simulations ...................................... 55

4.1.3 Effect of Curvature ........................................................................... 56
4.1.3.1 Effect of Curvature-Experimental Work ............................................. 56
4.1.3.2 Effect of Curvature-Predicting Curvature ....................................... 57

4.1.4 Effect of Rotation-Predictive Capabilities ........................................ 58

4.1.5 Numerical Simulations of Combined Effects Concerning Predictive Capabilities ................................................................. 59

4.2 Work with Reference to Parallel Mode Rotation ..................................... 61

CHAPTER FIVE: NUMERICAL APPROACH AND BOUNDARY CONDITIONS ... 65

5.1 Discretization ..................................................................................... 65
5.1.1 Second-Order Upwind Scheme ......................................................... 67
5.1.2 Discretization of the Momentum Equation ..................................... 68
5.1.3 Pressure Interpolation Schemes ....................................................... 68
5.1.4 Discretization of the Continuity Equation ...................................... 69
5.1. 5 Density Interpolation Schemes ................................................................. 70
5.1. 6 Pressure-Velocity Coupling .................................................................... 70
5.1. 7 Cell-Based Derivative Evaluation ............................................................. 71
5.1. 8 Multigrid Method ..................................................................................... 71

5. 2 Grid Generation and Boundary Conditions for Channels in Orthogonal Mode Rotation
........................................................................................................................................ 72
5.2. 1 Rib-Roughened Channel – Stationary Case ............................................. 72
5.2. 2 Smooth-Walls Channel – Rotating Case .................................................. 74
5.2. 3 Rib-Roughened Channel – Rotating Case .............................................. 77

5. 3 Grid Generation and Boundary Conditions for Channel in Parallel Mode Rotation 81

CHAPTER SIX: RESULTS AND DISCUSSION ........................................................... 83
6. 1 Comparison Between Two-Equation EVMs and RSM Turbulence Models ....... 83
6.1. 1 Stationary Rib-Roughened Channel .......................................................... 83
6.1. 2 Rotating Rib-Roughened Channel ............................................................ 86
6.1. 3 Discussion on the Performance of Turbulence Models and the Assumptions Involved
......................................................................................................................................... 94
6.1.3. 1 Assumptions Involved in Closing Two-Equation Models...................... 94
6.1.3. 2 Closure Coefficients Determination ....................................................... 96
6.1.3. 3 Flows with Adverse Pressure Gradient .................................................. 96
6.1.3. 4 Advantages of RSM with Enhanced Near Wall Treatment ................. 96

6. 2 Effect of Coriolis and Centrifugal Forces on Turbulence and Transport at High Rotation
Numbers and Density Ratios in Internal Cooling Channels with Smooth Walls – Results Using
Standard Wall Functions .............................................................................................. 97
6.2. 1 Effect of Combined High Rotation and Density Ratio on Nusselt Number... 97
6.2. 2 Comparison Between Results Using Standard Wall Functions and Enhanced Near Wall Treatment Approaches................................................................. 100
6. 3 Effect of Coriolis and Centrifugal Forces on Turbulence and Transport at High Rotation Numbers and Density Ratios in Internal Cooling Channels with Smooth Walls – Results Using Enhanced Near Wall Treatment Approach ................................................................. 103
  6.3. 1 Effect of High-Rotation Number on Nusselt Number ......................... 103
  6.3. 2 Velocity and Temperature Profiles ..................................................... 105
  6.3. 3 Secondary Flow .................................................................................. 109
  6.3. 4 The Influence of Fij on Dynamics ........................................................ 111
     6.3.4. 1 Predicted Reynolds Shear Stress Components ............................ 114
     6.3.4. 2 Normal Reynolds Stress Components .......................................... 117
  6.3. 5 Local Nusselt Number in the U-turn Region ........................................ 120
  6.3. 6 Average Nusselt Number in the U-turn region .................................... 121
  6.3. 7 Total Pressure Drop ......................................................................... 122
6. 4 Effect of Coriolis and Centrifugal Forces on Turbulence and Heat Transfer at High Rotation Numbers and Density Ratios in a Rib-Roughened Internal Cooling Channel 123
  6.4. 1 Temperature Distribution, Streamwise Velocity and Secondary Flows.... 124
  6.4. 2 Turbulence Anisotropy ...................................................................... 127
  6.4. 3 Reynolds Shear Stress Components .................................................. 129
  6.4. 4 Nusselt Number ................................................................................. 131
  6.4. 5 The Brunt-Vaisala Frequency ............................................................. 134
  6.4. 6 Total Pressure Drop .......................................................................... 137
LIST OF FIGURES

Figure 1. 1 Three Dimensional View of Turbine Blade and Platform Showing Cooling Systems........................................................................................................................................................................ 5
Figure 1. 2 Internal Cooling Turbine Blade Configuration ........................................ 6
Figure 1. 3 Schematic View of a U-bend Internal Cooling Channel in the Interior of Turbine Blade ........................................................................................................................................................................ 7
Figure 1. 4 Internal Cooling Channels Geometries in Copper Winding of Rotor Slot....... 9
Figure 1. 5 Schematic View of a Square Sectioned Internal Cooling Channel, Located inside Copper Winding of Generator Rotor ........................................................................................................ 9
Figure 3. 1 Velocity Contours and Velocity Vectors around a Rib Showing Re-Circulating bubble. RSM Predictions with Enhanced Wall Treatment ......................... 37
Figure 3. 2 Secondary Flow Vectors of First and Second Type. Ro=0.475, Re=25,000, DR=0.13, RSM Predictions. ................................................................. 41
Figure 3. 3 Secondary Flows at the Center (Left) and Exit (Right) of Square Cross Section U-bend - Stationary Case, Re=25,000, DR=0.13, RSM Predictions. ................. 43
Figure 5. 1 Control Volume Used to Illustrate Discretization of a Scalar Transport Equation ................................................................................................................................. 67
Figure 5. 2 Numerical Grid for Stationary Channel with in-Line ribs Studied Experimentally by Rau et al. [49]........................................................................................................ 73
Figure 5.3 Geometry for Two Pass Square Channel Tested Experimentally by Wagner et al. [74] ................................................................. 75

Figure 5.4 Numerical Grid for Channel with Smooth Wall Studied Experimentally by Wagner et al. [75] ................................................................................................................................. 76

Figure 5.5 Predicted and Measured, [74] Nusselt Number Ratios ...................... 77

Figure 5.6 Geometry for Two Pass Square Channel with Staggered Ribs Studied Experimentally by Wagner et al. [85] ........................................................................................................... 79

Figure 5.7 Rib Geometry and Numerical Grid for Rib-Roughened Rotating Channel ... 80

Figure 5.8 Predicted and Measured, [85] Nusselt Number Ratios on Leading Surface;
Re=25,000, DR=0.13 .................................................................................................................. 80

Figure 5.9 Numerical Grid for Square Channel in Parallel Mode Rotation ............. 82

Figure 6.1 Heat Transfer Enhancement along a Vertical Line on the Smooth Wall-Stationary Case at Distance e from the Last Rib................................................................. 84

Figure 6.2 Heat Transfer along the Centerline of the Floor in Between the Ribs-Stationary Case ................................................................................................................................. 84

Figure 6.3 Streamwise Velocity at Y/e = 0.1 in Symmetry Plane ......................... 85

Figure 6.4 Flow Entrainment in Between the Ribs at Y/e = 1 in Symmetry Plane ....... 85

Figure 6.5 Predicted and Measured, [85] Nusselt Number Ratios on Leading Surface; DR=0.13. ............................................................................................................................. 86

Figure 6.6 Centerline Local Nusselt Number on the Leading Surface. Ro=0.238, DR=0.13. ................................................................................................................................. 87

Figure 6.7 Centerline Local Nusselt Number on the Trailing Surface. Ro=0.238, DR=0.13. ................................................................................................................................. 87
Figure 6. 8 Streamwise Velocity Profile. Ro=0.238, DR=0.13, Re=25,000 ............... 88

Figure 6. 9 Turbulence Anisotropy in Terms of w’w’/v’v’ Contours in the Centerline of the First and Second Passes and in Terms of u’u’/v’v’ in the U-turn. Ro=0.238, DR=0.13, Re=25,000. Predicted Using RSM ................................................................. 90

Figure 6. 10 Secondary Flow vectors (colored by streamwise velocity) in First Pass (S/Dh = 12.4), U-turn and Second Pass. Ro=0.238, DR=0.13, Re=25,000 ............................... 93

Figure 6. 11 Nusselt Number Ratio in the First Pass .................................................. 98

Figure 6. 12 Nusselt Number in the U-turn .................................................................. 99

Figure 6. 13 Nusselt Number in the Second Pass ......................................................... 99

Figure 6. 14 Reynolds Normal Stress, W’W’ Using Standard Wall Functions (top) and Enhanced Near Wall Treatment (bottom) .............................................................. 101

Figure 6. 15 Reynolds Shear Stress, V’W’ Using Standard Wall Functions (top) and Enhanced Near Wall Treatment (bottom) .............................................................. 102

Figure 6. 16 Nusselt Number Ratio on the Leading and Trailing Surfaces ................. 104

Figure 6. 17 Streamwise Velocity Profile in Vertical Direction .................................. 106

Figure 6. 18 Streamwise Velocity Profile in Horizontal Direction ............................. 107

Figure 6. 19 Temperature Distribution ...................................................................... 108

Figure 6. 20 Secondary Flow Vectors ........................................................................ 110

Figure 6. 21 Reynolds Shear Stress Components in Vertical Direction (Between Trailing and Leading Surfaces) ................................................................. 115

Figure 6. 22 Reynolds Shear Stress Components in Horizontal Direction (Between Inner and Outer Surfaces) ................................................................. 116

Figure 6. 23 Stream Wise Normal Stress Components in Vertical Direction ............ 118
Figure 6. 24 Stream Wise normal Stress Components in Horizontal Direction ........... 119
Figure 6. 25 Average Nusselt Number in the U-turn Region for DR=0.13 ............... 122
Figure 6. 26 Total Pressure Drop-Smooth Walls Channel ....................................... 123
Figure 6. 27 Streamwise Velocity Vectors (W and U) and Temperature (θ) Contours in First Pass (Left), U-turn (Center) and Second Pass (Right) .................................................... 125
Figure 6. 28 Secondary Flow Vectors and Dimensionless Temperature (θ) Contours . 126
Figure 6. 29 Secondary Flow Vectors and Dimensionless Temperature (θ) Contours (Continued) ................................................................................................................................. 127
Figure 6. 30 Turbulence Anisotropy in Terms of W’W’/V’V’ Contours in the First and Second Pass and in Terms of U’U’/V’V’ in the U-turn .................................................. 129
Figure 6. 31 Reynolds Shear Stresses (Z/Dh=0 Corresponds to Inner Surface in the U-Turn) ................................................................................................................................. 130
Figure 6. 32 Reynolds Shear Stresses in Terms of V’W’/Wo^2 Contours in the First and Second Pass and in Terms of U’V’/Wo^2 in the U-turn . (Values Shown are the +ve and –ve maximum) ................................................................................................................................. 131
Figure 6. 33 Effect of Increasing Ro and DR on Local Nu/Nuo on Leading and Trailing Surfaces ................................................................................................................................. 132
Figure 6. 34 Effect of Increasing Ro and DR on Local Nu/Nuo on Inner and Outer Surfaces ................................................................................................................................. 133
Figure 6. 35 Brunt Vaisala Frequency at the Center of the First Pass. The Flow Direction is from Left to Right .................................................................................................................... 136
Figure 6. 36 Brunt Vaisala Frequency at the Center of the Second Pass. The Flow Direction is from Right to Left .................................................................................................................... 137
Figure 6.37 Mass Weighted Total Pressure Drop – Rib-Roughened Channel .......... 138
Figure 6.38 Calculated Average Nu and Measured Local Nu (Morris [131]) ............ 140
Figure 6.39 Nusselt Number Distribution on the Left Surface .................................. 141
Figure 6.40 Nusselt number Distribution on the Right Surface ............................... 141
Figure 6.41 Nusselt Number Distribution on the Bottom Surface ......................... 142
Figure 6.42 Nusselt Number Distribution on the Top Surface ............................... 143
Figure 6.43 Secondary Flow Vectors Colored by Streamwise Velocity ................. 145
Figure 6.44 Reynolds Shear Stress Components (m²/s²) at Z/L=0.8 (Y/Dh=0 corresponds to bottom surface and Y/Dh=1 corresponds to top surface) ................................. 147
Figure 6.45 Reynolds Shear Stress (m²/s²) Components in Between Z/L=0.8 and Z/L=0.9148
Figure 6.46 Reynolds Normal Stress Components (m²/s²) at Z/L=0.8 (Y/Dh=0 corresponds to bottom surface and Y/Dh=1 corresponds to top surface) ................................. 149
Figure 6.47 Turbulence Anisotropy in Terms of W’W’/V’V’ Contours in Between Z/L=0.8 and Z/L=0.9 ......................................................... 150
Figure 6.48 Mass Weighted Average Total Pressure Drop – Parallel Mode Rotation .. 151
Figure 8.1 Schematic Layout of Proposed Experimental Set-up ............................ 161
Figure 8.2 Schematic for Data Acquisition System .............................................. 161
LIST OF TABLES

Table 2. 1 Two-Equation \( k-\varepsilon \) Turbulence Models .......................................................... 21
Table 2. 2 Two-Equation \( k-\omega \) Turbulence Models ......................................................... 23
Table 2. 3 Reynolds Stress Turbulence Model ................................................................. 26
Table 3. 1 Near-wall Treatment using Standard Wall Functions ........................................ 34
Table 3. 2 Enhanced Near Wall Treatment using Combined Two Layer Model with Enhanced Wall Functions ........................................................................................................ 35
Table 4. 1 Experimental Investigations on Heat transfer and Turbulence in Parallel Mode Rotating Ducts. ................................................................................................................. 62
Table 6. 1 The Influence of \( F_{ij} \) on Dynamics in Orthogonal-Mode Rotation ............... 112
Table 6. 2 Local Nusselt Number in the U-turn Region ...................................................... 121
Table 6. 3 The Influence of \( F_{ij} \) on Dynamics in Parallel-Mode Rotation ...................... 139
## LIST OF ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Full Form</th>
</tr>
</thead>
<tbody>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>DR</td>
<td>Density Ratio</td>
</tr>
<tr>
<td>EARSM</td>
<td>Explicit Algebraic Reynolds Stress Model</td>
</tr>
<tr>
<td>EVM</td>
<td>Eddy Viscosity Model</td>
</tr>
<tr>
<td>GGDH</td>
<td>Generalized Gradient Diffusion Hypothesis</td>
</tr>
<tr>
<td>HRN</td>
<td>High Reynolds Number</td>
</tr>
<tr>
<td>IGV</td>
<td>Inlet Guide Vanes</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LRN</td>
<td>Low Reynolds Number</td>
</tr>
<tr>
<td>P</td>
<td>Pitch</td>
</tr>
<tr>
<td>PRESTO</td>
<td>PREsure STaggering Option</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Average Navier Stokes</td>
</tr>
<tr>
<td>RNG</td>
<td>Renormalization Theory Group</td>
</tr>
<tr>
<td>RPM</td>
<td>Rotation Per Minute</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds Stress Model</td>
</tr>
<tr>
<td>SFC</td>
<td>Specific Fuel Consumption</td>
</tr>
<tr>
<td>SGDH</td>
<td>Simple Gradient Diffusion Hypothesis</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Implicit Method for Pressure-Linked Equations</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
</tbody>
</table>
TET  Turbine Entry Temperature
LIST OF VARIABLES

\( A_f \)  Area of face

\( a_{ce} \)  Acceleration due to centrifugal force

\( a_{co} \)  Acceleration due to Coriolis force

\( C_f \)  Friction factor

\( C_0 \)  Cell number 0

\( C_1 \)  Cell number 1

\( De \)  Dean number

\( DR \)  Density ratio

\( D_h \)  Hydraulic diameter

\( E_o \)  Total internal energy

\( e \)  Rib height

\( f_{ce} \)  Centrifugal force

\( f_{co} \)  Coriolis force

\( Gr \)  Grashoff number

\( h \)  Heat transfer coefficient

\( J_f \)  Mass flux at face \( f \)

\( J_b \)  Rotational Reynolds number = \( \Omega b^2/2\nu \)

\( k \)  Thermal conductivity of coolant (w/m \( \cdot \) C)
Kₘₙ Nikuradse’s roughness height
Nu Local Nusselt number, hDₘₙ/k
Nuₒ Nusselt number in fully developed turbulent nonrotating tube flow
Nur Calculated Nusselt number for Ro>0
Nus Calculated Nusselt number for Ro=0
Pr Prandtl number
Prₜ Turbulent Prandtl number
R Radius from axis of rotation
S Distance in streamwise direction
T Local coolant temperature
R Radius from axis of rotation
Re Reynolds number, ρWₒDₘₙ/μ
r Inner radius of bend
Ro Rotation number, ΩDₘₙ/Wₒ
T₁ The inlet temperature
T₃ The turbine entry temperatures (TET)
Tₜ Coolant bulk temperature
Tₒ Coolant temperature at inlet
Tₚ Wall temperature
V Volume of cell
Wₒ Inlet velocity
Greek

\( \phi \)  
Scalar quantity

\( \Gamma_\phi \)  
Diffusion coefficient

\( \eta \)  
Carnot efficiency

\( \theta \)  
Dimensionless temperature, \( (T-T_o)/(T_w-T_o) \)

\( \mu \)  
Dynamic viscosity of coolant

\( \rho \)  
Density of air

\( \Delta \rho / \rho \)  
Density ratio, \( (T_w-T_o)/T_w \)

\( \Omega \)  
Rotational speed in RPM

\( \omega \)  
Rotational speed in Rad/Sec

Subscript

b  
Bulk

ce  
Centrifugal

c, ,  
Coriolis

f  
Cell face

h  
Hydraulic

o  
Inlet

r  
Rotating

s  
Stationary

t  
Turbulent

w  
Wall
CHAPTER ONE: INTRODUCTION

1.1 Background

Sustainable and cheap sources of energy and water are two of the most important issues for the society of tomorrow. In this aspect turbines and generators, the workhorses that provide energy to millions of houses and factories will continue to remain important tomorrow.

High-speed rotating machines such as gas turbines and heavy-duty electrical generators must be designed for high reliability as any failure in these machines may lead to catastrophic failure in the systems that they support. This is particularly true for generators and turbines in an aircraft. Reliability of any generator and turbine is critically dependent on the temperature rise in their individual components while in operation. For example, it has been observed that every 7-8°C rise in operating temperature of electrical machines beyond the peak design value reduces the life of their insulation to half. Similarly the creep life of turbine blades is also reduced to half with every 10 to 15°C rise in metal temperature. In other words, hot components design of these machines must aim to keep their operating temperatures within some tolerable limits. In order to keep the operating temperatures of these hot components under a prescribed limit, they are cooled with a combination of advanced cooling techniques. However, one common technique
that is used for both turbine blades and generator rotors is to have a coolant fluid flowing through internal cooling channels formed within the hot components.

For decades researchers have investigated various aspects of internal cooling in both generators and turbines in order to understand flow and heat transfer characteristics in these cooling channels so that more optimal and efficient designs can be achieved. One of the particularly difficult aspects of better understanding the associated physical processes has been to understand how turbulence production, dissipation and transport are affected by rotation inside rotating internal cooling channels used in turbine blades and generator rotors. The basic philosophy of cooling techniques employed in the turbine blades and generator rotors are similar, but different in term of the relative direction of coolant flow with respect to the respective axis of rotation. In gas turbine blades the internal cooling channels are primarily orthogonal to the axis of rotation (orthogonal-mode rotation), while in generator rotors, the cooling channels are parallel to the axis of rotation (parallel-mode rotation). Moreover, generator rotors and turbine blades use oil and air as coolants, respectively, which have very different properties that can potentially affect turbulence production, dissipation and transport.

1.2 Internal Cooling of Turbine Blades

The maximum allowable turbine inlet temperature strongly affects the efficiency of gas turbine engines and substantial performance increases can be achieved by increasing this temperature. The current industry trend is to increase the turbine inlet temperature closer to the fuel stoichiometric temperature, especially for military engines. High fuel costs and environmental
awareness are a big challenge for turbine designers to enhance the efficiency. The specific fuel consumption (SFC) and the specific work output are important to determine efficiency and effectiveness. Carnot efficiency is a measure of the maximum theoretical efficiency of a gas turbine cycle and is given by:

\[ \eta = 1 - \frac{T_1}{T_3} \]  

(1.1)

where

\( \eta \) is Carnot efficiency

\( T_1 \) is the inlet temperature

and

\( T_3 \) is the turbine entry temperatures (TET).

Thus increasing TET improves the efficiency. However, there are many factors that have a decreasing effect on efficiency, pressure and mass flow losses, non-ideal fluids, friction, components efficiency, etc. As a result of these losses the compressor pressure ratio affect the efficiency of a simple cycle as well. Thus both high TET and high-pressure ratio are required to enhance the efficiency. The high TET could cause a failure in the structure of the hot components, which are made of high strength materials such as nickel and cobalt based super alloys, (eg. Inco 738 and Rene 220). The engine life could be extended through the introduction of relatively cool gas from the compressor in places of highest temperatures in turbine. The highest temperature loads are at the exit of the combustor, and in the first turbine stage. An efficient cooling system is thus needed for the inlet guide vanes (IGV). External cooling (film cooling), and internal cooling (convection- and impingement-cooling) are usually used to cool
these IGV. Film cooling is also applied to the platforms, along the tip (shroud) and root (hub) of the vane. In the internal side, convection and impingement cooling techniques are used, where the compressed air is supplied through channels, which cools the vanes by means of conduction/convection from the inside. Ribs, arranged orthogonal to the flow or angled ribs, are introduced in the channels, which makes the flow separate and re-attach with an increase in turbulence level and consequently enhancement of heat transfer. Then the cooling air is flowing vertically towards some certain hot regions to provide impingement cooling. The available pressure difference between the internal cooling air and the external main gas flow is extremely limited. Hence there is a restriction to the number of turns and ribs that can be employed inside a passage. There are also limitations to the complexity and cost of the internal cooling channel.

Despite the significant advantage of increasing TET, the use of cooling air for achieving this has some disadvantages: 1) the addition of cool air into the mainstream reduces the work output from the turbines; 2) protective films along the vanes complicates the aero-thermal design of the blades, as the momentum and the blockage effect introduced by cooling air changes gas angles dependent on engine loads; 3) cooling air does not participate in the energy enriching process in combustor and hence the effective mass flow is reduced.

The cooling technique currently used for blades of high pressure turbines is a combination of internal and film cooling. In this technique, cooler air is injected into serpentine passages within the blade. Most of this air issues out of tiny (film-cooling) holes into the high temperature boundary layer on the blade surface, in an effort to form a cooler layer between the hot gas stream and the blade surface, see Figure 1.1 and Figure 1.2.
Figure 1. 1 Three Dimensional View of Turbine Blade and Platform Showing Cooling Systems
The blades, in addition to the temperature loads are however stressed by the rotational forces, (ie. the Coriolis and the centrifugal forces). These rotational induced forces complicate the flow structure within the channel. Figure 1.3 shows a schematic view of a U-bend section, located in the center of turbine blade. The forces acting on the flow field and the boundary conditions are shown. The effects of channel walls, corners, bends, ribs, heating and rotation characterize the flow phenomena inside internal cooling channels of turbine blades. These effects and their influence on the flow are to be addressed in the following chapters.
Inward flow

Secondary flow

Figure 1.3 Schematic View of a U-bend Internal Cooling Channel in the Interior of Turbine Blade

1.3 Internal Cooling of Rotors of Electrical Machines

Generators that are used in aircrafts rotate at high speed and turbine-driven generators that produce electrical power have a relatively high output per machine. The performance of these machines depends on stress, rotor vibrations and thermal behavior. Once the physical size of the machine has been limited by the mechanical constraints, the only remaining way by which the power output may be increased is via increases in the electrical and magnetic loadings in the
stator and rotor of the machine. The implied consequential increase in the absolute level of
general inefficiencies within the machine necessitates a reliable cooling system to be
incorporated into the fundamental design concept. This is to ensure that the thermal losses are
dissipated at temperature levels compatible with an acceptable lifespan of the electrical
insulation materials used in generator construction.

The rotors of electrical machines are cooled by passing fluid inside the hollow conductors of
these machines. The shape of the coolant channels inside the windings of rotor can vary from
square to circular. Figure 1. 4 shows the windings of the rotor with their internal cooling
channels, mounted in slots machined axially along the rotor forging. The flow circuit geometries
selected for cooling rotor windings is closely related to the machine rating and choice of coolant
made. However, it typically involves a combination of bends, expansions, contractions, plenum
chambers, etc. In cooling of rotors of electrical machines, the coolant flows in channels parallel
to the rotational axis (parallel-mode rotation).

Figure 1. 5 shows a schematic view of a square sectioned internal cooling channel, located inside
copper winding of generator rotor. The Coriolis and centrifugal forces that are acting on the
flow field and the boundary conditions are shown.
1.4 Problem Statement

It is proposed to perform flow field and heat transfer study for internal cooling channels of generator rotors and turbine rotor blades that rotate at high rotation numbers (up to 1.29) and high-density ratio (up to 0.4) using the current state-of-the-art, practically viable computational methods.
approach. Particular attention would be given to how Reynolds stress, turbulence intensity, and transport are affected by Coriolis and buoyancy/centrifugal forces caused by high levels of rotation and density ratio. The results obtained will be explained in view of physical interpretation of Coriolis and centrifugal forces. The tasks are defined as follows:

1. Studying turbulence models: two-equation models and RSM.

2. Using a commercial CFD code (FLUENT) to study the complex flow and heat transfer phenomena and employing enhanced near wall treatment to resolve the near wall viscous region.

3. Comparing the CFD results of two-equation and RSM turbulence models with experimental data available in open literature, which are mainly at low rotation numbers and low density ratios.

4. Using the turbulence model that performs best to study the following:
   - Effect of high-rotation number and density ratios on flow and heat transfer in square internal cooling cannel with smooth walls and U-turn and with rib-roughened walls in orthogonal mode rotation.
   - Effect of high-rotation number and density ratio on flow field and heat transfer in square internal cooling cannels with smooth walls in parallel mode rotation.
1.5 Methodology

The methodology to perform the above-required tasks is as follows:

1. Utilizing the employed CFD code such that it will be able to predict the complex flow and heat transfer phenomena inside rotating internal cooling channels. In particular the code should be able to predict:

   - Anisotropy, separation, acceleration-deceleration and shearing in channels with U-bend and ribs.
   - Turbulent induced Secondary flows formed in channel corners, ribs induced and Coriolis induced secondary flows.
   - High level of Coriolis and buoyancy forces caused by rotation and heating.

2. Verification of results should base on experimental comparison.

   Experimental data is available only at low rotation numbers and density ratios. Thus results for cases with high rotation numbers and density ratios must be reasonable and explained in view of Coriolis and centrifugal forces interpretation.

3. The results from this study, should be applicable to the actual internal cooling channels geometry and operating conditions to derive flow and heat transfer correlations for practical applications.
1. 6 Scope

The scope of this research is to study the flow field and heat transfer in internal cooling channels at high rotation numbers and density ratios using accurate turbulence model that requires relatively low computational cost.

The appropriate approach that would ensure relatively accurate results at a moderate computational cost would be outlined. The results obtained would be validated against available experimental results. The methodology and results from this study, once validated against appropriate high-speed experimental results in the future, can be applied to the actual operating conditions to derive flow and heat transfer correlations for practical applications.

1. 7 Uniqueness of the Research and its Contribution

There are several aspects of this dissertation that will contribute to new knowledge:

A comprehensive study of two-equation and RSM turbulence models capabilities to predict the combined effect of high rotation and buoyancy numbers flow and heat transfer in internal cooling channels.

Studying internal cooling channels with high rotation and buoyancy numbers is unique because the existing knowledge concentrating on low rotation and buoyancy numbers.
Research on flow and heat transfer in parallel mode rotating internal cooling channels is a contribution to new knowledge, which is hard to find in open literature.
CHAPTER TWO: GOVERNING EQUATIONS, TURBULENCE AND HEAT TRANSFER MODELS

2.1 Governing Equations

Navier-Stokes equations that govern fluid motion and heat transfer are continuity, momentum, and energy. These equations are given in this chapter in conservative form.

2.1.1 Continuity Equation

The continuity equation for all but nuclear-reaction environments is valid. It states the conservation of mass and in conservative form it is given by:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0$$

(2.1)

2.1.2 Momentum Equation

The momentum equation is a force balance derived from Newton’s second law, which states that mass times acceleration is equal to imposed forces. There are two types of forces, body forces
\( (F_i) \), eg. the gravitational force, and surface forces, \( (T_{ij}) \). The surface forces are given by:

\[
T_{ij} = -p\delta_{ij} + \tau_{ij}
\]

where the first term is the pressure (normal stress) and the second term is the viscous stress (shear). For Newtonian incompressible fluid the momentum equations becomes:

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial x_i} = \rho g_i + F_i - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i} (2\mu S_{ij})
\]

\[ (2.2) \]

where \( F_i \) are the additional body-forces such as rotation, magnetic or electric field etc and \( S_{ij} \) is the strain rate tensor.

2.1.3 Energy Equation

From the first law of thermodynamics, the exchange of energy for a system is the result of applied work and heat transfer through that system. The energy equation, Burmeister [2] is given as:

\[
\frac{\partial \rho E_0}{\partial t} + \frac{\partial \rho U_i E_0}{\partial x_i} = \rho U_i F_i - \frac{\partial q_i}{\partial x_j} + \frac{\partial}{\partial x_j} (U_j T_{ij})
\]

\[ (2.3) \]

where \( T_{ij} \) are the surface forces similar to the viscous and pressure terms in the momentum equations. \( E_0 \) is the total internal energy.

2.1.4 Acceleration Terms due to Rotation

The Navier-Stokes equations discussed above did not include the effect of rotation. In rotating coordinate frame the Coriolis and centripetal accelerations are involved and need to be account
for. Assuming that the coordinate system has a fixed origin and that the rotational velocity is constant, the additional body-forces (e.g. Coriolis and centripetal accelerations) that appear on LHS in the Navier-Stokes momentum equation are given as follows:

\[
a_{i}^{ce} = \varepsilon_{ijk} \Omega_{k} \Omega_{j} x_{m}
\]

(2.4)

\[
a_{i}^{co} = 2\varepsilon_{ijk} \Omega_{j} U_{k}
\]

(2.5)

2.2 Turbulence Models

The Navier-Stokes equations are non-linear partial differential equations that cannot be solved analytically, except a few special cases. Consequently, for most cases, numerical solution is needed to solve the NS-equations. Direct Numerical Simulations (DNS’s) and Large Eddy Simulations (LES’s) are the most accurate but they require large computational resources. Reynolds Averaged Navier Stokes (RANS) turbulence models although are affected by numerical and physical approximations still they perform reasonably accurate, require less computational resources and they are widely used for industrial applications.

2.2.1 Direct Numerical Simulations

Direct Numerical Simulations (DNS’s) solve the equations in a four-dimensional domain, time and space. DNS is a time accurate and can resolve all length scales. The major disadvantage is that the simulations are computational expensive and can only be performed for a limited number of flows with relatively low Reynolds numbers. To illustrate this, consider the length scale ratio \( L / \eta \), where \( L \) is the integral length scale, and \( \eta \) is the Kolmogorov length scale defined as:
\[ \varepsilon \sim \frac{U^3}{L} \quad \text{and} \quad \eta = \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \]

thus \[ \frac{L}{\eta} \sim R_{t}^{3/4} \], with the assumption that \( (L/\eta)^3 \) is proportional to the number of grid points, the computational grid increases as: \( R_{t}^{9/4} \). As Reynolds number increases the time step decreases and thus the computational time increases rapidly.

### 2.2.2 Large Eddy Simulation (LES) Model

Turbulent flows are characterized by eddies with a wide range of length and time scales. The largest eddies are typically comparable in size to the characteristic length of the mean flow. The smallest scales are responsible for the dissipation of turbulence kinetic energy. Basically large eddies are resolved directly in LES, while small eddies are modeled. The rationale behind LES can be summarized as follows:

Momentum, mass, energy, and other passive scalars are transported mostly by large eddies. Large eddies are more problem-dependent. They are dictated by the geometries and boundary conditions of the flow involved. Small eddies are less dependent on the geometry, tend to be more isotropic, and are consequently more universal. Solving only for the large eddies and modeling the smaller scales results in mesh resolution requirements that are much less restrictive than with DNS. Furthermore, the time step sizes will be proportional to the eddy-turnover time, which is much less restrictive than with DNS. However, extremely fine meshes are still required. LES is situated somewhere between DNS and the RANS approach.
2.2.3 Reynolds Average Navier Stokes Models, (RANS)

According to Reynolds [3] the quantities in the NS-equation could be divided into mean and fluctuating components:

\[ \phi = \bar{\phi} + \phi' \]  

(2.6)

where the mean component is the time-average of a parameter over time. Applying Reynolds approach to NS-equations results in the Reynolds Averaged Navier-Stokes equations (see White [4] for details). This procedure produces the Reynolds stresses: \( \tau_{ij} = \bar{u}_i \bar{u}_j \), which are unknown and need to be modeled using turbulence model. This is referred to as the closure problem with Reynolds averaging. To model the Reynolds stresses there are two approaches, the eddy-viscosity models (EVM), and the Reynolds stress models (RSM). In RSM the actual stresses are solved, while in the EVM the Boussinesq hypothesis is employed to estimate \( \tau_{ij} \).

### 2.2.3.1 Eddy-Viscosity Models (EVMs)

To estimate the Reynolds stresses, \( \tau_{ij} \), in the EVM the Boussinesq hypothesis approach is employed.

#### 2.2.3.1.1 Boussinesq Hypothesis

Using a concept similar to the molecular viscosity for molecular stresses, the concept of the eddy viscosity may be used to model the Reynolds stresses. For general flow situations the eddy viscosity model may be written as:
\[ -u_i u_j = v_t(x, y, z) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \]  

(2.7)

where \( v_t \) is the eddy-viscosity and \( k \) is the turbulent kinetic energy. The second part can be absorbed into the pressure-gradient term so that instead of static pressure an unknown modified pressure given as: \( p^\prime = p + 2/3 \rho k \). Therefore, the main problem in this concept is to determine the distribution of \( v_t \).

The turbulent heat transfer is treated in direct analogy to the turbulent momentum transport assuming to be related to the gradient of the transported quantity, with eddies again replacing molecules. Therefore, the Reynolds flux terms are given as:

\[ -u_i \bar{\theta} = \Gamma_i \frac{\partial \bar{\theta}}{\partial x_i} \]  

(2.8)

where \( \Gamma_i \) is the turbulent diffusivity of heat and it is usually given by \( \Gamma_i = v_t / \sigma \), where \( \sigma \) is the turbulent Prandtl number, which is constant approximately equal to one.

In EVM the directional properties (anisotropic) are not considered and the turbulence is assumed isotropic. While this is true for smaller eddies at high Re, the large eddies are anisotropic due to the strain rate of the mean flow. The local equilibrium assumption in EVMs is another problem where the production is assumed to be equal locally to the dissipation term. For uni-directional flow and for flow where turbulence is evolving at a sufficiently rapid rate, such that the effects of past events do not dominate the dynamics, the estimates based on local scales can give relatively accurate results. Typical flows where two-equation models have been shown to fail are flows
with sudden changes in mean strain rate, curved surfaces, secondary motions, rotating and stratified fluids, flows with separation, and three-dimensional flows.

2.2.3.1.2 Two-Equation EVMs

A number of two-equation EVMs exists in literature, in this dissertation the following EVMs are considered (which are available in FLUENT [5] CFD Code):

- The Standard $k-\varepsilon$ Model, Launder and Spalding [6]
- The Renormalization Group $k-\varepsilon$ model by [7]
- The Realizable $k-\varepsilon$ model by [8]
- The Standard $k-\omega$ Model, Wilcox [9]
- The Shear-Stress Transport (SST) $k-\omega$ Model by Menter [10]

In $k-\varepsilon$ models the turbulence kinetic energy, $k$, and its rate of dissipation, $\varepsilon$, are obtained from the following general transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho ku_i) = \frac{\partial}{\partial x_j} \left[ M_2 \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon + S_k \tag{2.9}$$

and

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ M_2 \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_1 - C_{2\varepsilon} G_2 + S_\varepsilon - R_\varepsilon + G_3 \tag{2.10}$$

Table 2.1 shows how the different terms in Equations 2.9 and 2.10 are modeled for each of the three $k-\varepsilon$ models.
Table 2. 1 Two-Equation $k-\varepsilon$ Turbulence Models

<table>
<thead>
<tr>
<th></th>
<th>Standard $k-\varepsilon$ model</th>
<th>(RNG) $k-\varepsilon$ model</th>
<th>Realizable $k-\varepsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>$\mu + \frac{\mu_t}{\sigma_k}$</td>
<td>$\alpha_k \mu_{eff}$</td>
<td>$\mu + \frac{\mu_t}{\sigma_k}$</td>
</tr>
<tr>
<td>$G_k$</td>
<td>Exact $= -\rho \frac{\partial u_i}{\partial x_j}$, based on Boussinesq hypothesis $= \mu_t \sqrt{2S_{ij}S_{ij}}$</td>
<td>= $\beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i}$</td>
<td></td>
</tr>
<tr>
<td>$Pr_t$</td>
<td>0.85</td>
<td>1/$\alpha$, varies by RNG</td>
<td>0.85</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>NA</td>
<td>Analytical formula by RNG</td>
<td>NA</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>$\rho C_\mu \frac{k^2}{\varepsilon}$ based on local equilibrium assumption</td>
<td>$\frac{d}{\sqrt{\varepsilon}} \frac{\mu g_i}{\mu} = 1.72 \frac{\mu g_i}{\mu} \frac{d\mu_{eff}}{d\mu}$</td>
<td>$\rho C_\mu \frac{k^2}{\varepsilon}$</td>
</tr>
<tr>
<td>$G_b$</td>
<td>$G_k + C_{g_b} G_h$</td>
<td>$C_{g_b} G_h$</td>
<td></td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>0.09, exper. (Weakness)</td>
<td>0.0845, RNG theory, Weakness</td>
<td>$f(S, \Omega, \omega, k, \varepsilon)$</td>
</tr>
<tr>
<td>M2</td>
<td>$\mu + \frac{\mu_t}{\sigma_\varepsilon}$</td>
<td>$\alpha_\varepsilon \mu_{eff}$</td>
<td>$\mu + \frac{\mu_t}{\sigma_\varepsilon}$</td>
</tr>
<tr>
<td>G1</td>
<td>$G_k + C_{g_b} G_h$</td>
<td>$C_{g_b} G_h$</td>
<td></td>
</tr>
<tr>
<td>G2</td>
<td>$\varepsilon^2 / k$, Causes singularity</td>
<td>$\varepsilon^2 / (k + \sqrt{ve})$</td>
<td></td>
</tr>
<tr>
<td>G3</td>
<td>NA</td>
<td>NA</td>
<td>$\rho C_1 S_\varepsilon$</td>
</tr>
<tr>
<td>$\sigma_k$</td>
<td>NA</td>
<td>Eq. accounts for rapid strain and streamline curvature</td>
<td></td>
</tr>
<tr>
<td>$\sigma_\varepsilon$</td>
<td>NA</td>
<td>NA</td>
<td></td>
</tr>
<tr>
<td>$C_1$</td>
<td>1.44, exper.</td>
<td>1.42 (analytic by RNG)</td>
<td>1.44</td>
</tr>
<tr>
<td>$C_2$</td>
<td>1.92, exper.</td>
<td>1.68 (analytic by RNG)</td>
<td>1.9 established</td>
</tr>
<tr>
<td>$\alpha_k$</td>
<td>NA</td>
<td>Varies by RNG</td>
<td>NA</td>
</tr>
<tr>
<td>$\alpha_\varepsilon$</td>
<td>NA</td>
<td>Varies by RNG</td>
<td>NA</td>
</tr>
<tr>
<td>Conv. Heat Transfer Modeling</td>
<td>$\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} [u_i (\rho E + p)] = \frac{\partial}{\partial x_j} \left( k_{eff} \frac{\partial T}{\partial x_j} + u_i (\tau_{ij})_{eff} \right) + S_h$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deviatoric stress tensor</td>
<td>$\left( \tau_{ij} \right)<em>{eff} = \mu</em>{eff} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_i}{\partial x_j} \delta_{ij}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$k_{eff}$</td>
<td>$k + \frac{C_p \mu_t}{Pr_t}$</td>
<td>$\alpha C_p \mu_{eff}$</td>
<td></td>
</tr>
<tr>
<td>$Pr_t$</td>
<td>0.85</td>
<td>0.85</td>
<td>Varies</td>
</tr>
</tbody>
</table>
In k-ω models the turbulence kinetic energy, $k$, and the specific dissipation rate, $\omega$, are obtained from the following general transport equations:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (2.11)$$

and

$$\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[ \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega + D_\omega \quad (2.12)$$

Table 2.2 shows how the different terms in Equations 2.11 and 2.12 are modeled for both models.
Table 2. Two-Equation k-ω Turbulence Models

<table>
<thead>
<tr>
<th>Description</th>
<th>Standard k-ω model</th>
<th>SST k-ω model</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Greek symbols</strong></td>
<td><strong>Empirical, based on model transport equations for (k) and (ω)</strong></td>
<td><strong>Modified turbulent viscosity, addition of a cross-diffusion term in the (ω) equation and a blending function.</strong></td>
</tr>
<tr>
<td>$\Gamma_k$ (Diffusivity)</td>
<td>$\mu + \frac{\mu_t}{\sigma_k}$</td>
<td></td>
</tr>
<tr>
<td>$\Gamma_\omega$</td>
<td>$\mu + \frac{\mu_t}{\sigma_\omega}$</td>
<td></td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>$\alpha^* \frac{\rho k}{\omega}$</td>
<td>$\rho k \max \left[ \frac{1}{\alpha^*}, \frac{\Omega F_2}{0.31 \omega} \right]$</td>
</tr>
<tr>
<td>$\alpha^*$ (LRN correc.)</td>
<td>$\alpha^* \frac{0.024 + \text{Re}_i / R_k}{1 + \text{Re}_i / R_k}$</td>
<td>where F is a blending function</td>
</tr>
<tr>
<td>$\alpha^*$ (HRN)</td>
<td>$1$</td>
<td></td>
</tr>
<tr>
<td>Re$_i$</td>
<td>$\rho k / \mu \omega$</td>
<td></td>
</tr>
<tr>
<td>$G_k$ (Production)</td>
<td>$\rho \mu \frac{\partial u_j}{\partial x_i}$</td>
<td></td>
</tr>
<tr>
<td>$G_\omega$</td>
<td>$\omega = \alpha \frac{\omega}{k} G_k$</td>
<td>$\alpha G_k / \nu_i$</td>
</tr>
<tr>
<td>$\alpha$ (LRN correc.)</td>
<td>$\frac{0.11 + \text{Re}_i / R_k}{1 + \text{Re}_i / R_k}$</td>
<td></td>
</tr>
<tr>
<td>$\alpha$ (HRN)</td>
<td>$1$</td>
<td></td>
</tr>
<tr>
<td>$\alpha_x$</td>
<td>$1$</td>
<td>$F_i \alpha_{x,i} + (1 - F_i) \alpha_{x,2}$</td>
</tr>
<tr>
<td>$Y_k$ (Dissipation)</td>
<td>$\rho \beta^* f_{\beta^*} k_{\omega}$</td>
<td></td>
</tr>
<tr>
<td>$f_{\beta^*}$</td>
<td>$\begin{cases} 1 &amp; \text{if } \omega &gt; 0 \ 0 &amp; \text{otherwise} \end{cases}$</td>
<td></td>
</tr>
<tr>
<td>$X_k$</td>
<td>$\frac{1}{1 + 680 X_k^2}$</td>
<td></td>
</tr>
<tr>
<td>$\beta^*$</td>
<td>$\beta_t^* \left[ 1 + \zeta^* F(M_t) \right]$ (LRN)</td>
<td>$\beta_t^* \left[ 1 + \zeta^* F(M_t) \right]$ (LRN)</td>
</tr>
<tr>
<td>$\beta_i^*$</td>
<td>$0.09 \left[ \frac{4/15 + \left( \text{Re}/8 \right)^4}{1 + \left( \text{Re}/8 \right)^4} \right]$ (LRN)</td>
<td>$0.09 \left[ \frac{4/15 + \left( \text{Re}/8 \right)^4}{1 + \left( \text{Re}/8 \right)^4} \right]$ (LRN)</td>
</tr>
<tr>
<td>$Y_\omega$</td>
<td>$\rho \beta^* \beta_t^* \omega^2$</td>
<td></td>
</tr>
<tr>
<td>( f_{\beta} )</td>
<td>( \frac{1+70X_{\omega}}{1+80X_{\omega}} )</td>
<td>1</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>( \beta )</td>
<td>( \beta_i \left[ 1 - \frac{\beta_i^*}{0.072} \right] 1.5F(M_t) )</td>
<td></td>
</tr>
<tr>
<td>( \beta_i )</td>
<td>0.072</td>
<td>( F_i \beta_{i,3} + 0.0828(1 - F_i) )</td>
</tr>
</tbody>
</table>
| \( F(M_t) \) | Compressible Correction | 0,...,\( M_t \leq 0.25 \) \[
\frac{2k}{\nu^2},...M_t > 0.25
\]
| \( D_\omega \) (Cross diffusion modif.) | NA | \( 2 \left( 1 - F_i \right) \rho \sigma_{w,2} \frac{1}{\omega} \partial k \partial \omega \)
| Wall B.C. | For HRN region : wall functions |
| \( \omega_w \) | \( \frac{\rho \left( u^* \right)^2}{\mu} \omega^+ \) | |
| \( \omega^+ \) (laminar sublayer) | \( \min \left( \omega_w^+, \frac{6}{0.09 \left( y^+ \right)^2} \right) \) | |
| \( \omega^+ \) | | |
| \( k_i^+ \) (Roughness height) | | |
| \( \omega^+ \) (Logarithmic region) | \( 3.33 \frac{d u^+_{lamb}}{d y^+} \) | |
| \( \omega \) | \( \frac{u^*}{0.3 k y} \) | |
| Buffer region | \( \omega^+ \) is blended between log. And laminar sublayer |

### 2.2.3 2 Reynolds Stress Models

Reynolds Stress Model (RSM) [11], [12], [13] solves the Reynolds Stresses, \( \tau_{ij} = u_i u_j \), using individual transport equations. The exact transport equations for the transport of the Reynolds stresses, may be written as follows:

\[
\frac{D \tau_{ij}}{Dt} = \frac{\partial \tau_{ij}}{\partial x_k} + \frac{\partial D_{ij}}{\partial x_k} + P_j + G_j + \phi_{ij} - \epsilon_{ij} + F_{ij} + S \quad (2.13)
\]
where the LHS is given by
\[
\frac{D\tau_{ij}}{Dt} = \frac{\partial}{\partial t}(\rho u_i u_j) + C_{ij}
\]
where \(\frac{\partial}{\partial t}(\rho u_i u_j)\) is the local time derivative, \(C_{ij}\) the convection term, \(D_{T,ij}\) the turbulent diffusion term, \(D_{L,ij}\) the molecular (viscous) diffusion term, \(P_{ij}\) the stress production term, \(G_{ij}\) the buoyancy production term, \(\phi_{ij}\) the pressure strain term, \(\varepsilon_{ij}\) the dissipation term, \(F_{ij}\) the production term by system rotation, and \(S\) the source term. Of the various terms in these exact equations, \(C_{ij}\), \(D_{L,ij}\), \(P_{ij}\), and \(F_{ij}\) do not require any modeling. However, turbulent diffusion (\(D_{T,ij}\)), buoyancy production (\(G_{ij}\)), pressure strain (\(\phi_{ij}\)), and dissipation (\(\varepsilon_{ij}\)) need to be modeled to close the equations.

Table 2.3 shows how the different terms in Equation 2.13 are modeled.
Table 2. 3 Reynolds Stress Turbulence Model

<table>
<thead>
<tr>
<th>RSM Model</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Description</strong></td>
<td>Calculation of $\tau_{ij} = \overline{u_i u_j}$, using differential transport equations, then $\tau_{ij} = \overline{u_i u_j}$ are used to obtain closure</td>
</tr>
<tr>
<td><strong>Convection</strong></td>
<td>$C_{ij} = \frac{\partial}{\partial x_k} \left( \rho \mu_k \overline{u_i u_j} \right) \quad \text{Exact}$</td>
</tr>
<tr>
<td><strong>Molecular (viscous) diffusion</strong></td>
<td>$D_{L,ij} = \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial}{\partial x_k} \left( \overline{u_i u_j} \right) \right] \quad \text{Exact}$</td>
</tr>
<tr>
<td><strong>Stress production</strong></td>
<td>$P_{ij} = -\rho \left( \overline{u_i u_k} \frac{\partial u_j}{\partial x_k} + \overline{u_j u_k} \frac{\partial u_i}{\partial x_k} \right) \quad \text{Exact}$</td>
</tr>
<tr>
<td><strong>Production by system rotation</strong></td>
<td>$F_{ij} = -2 \rho \Omega_k \left( \overline{u_i u_m e_{ikm}} + \overline{u_j u_m e_{jkm}} \right) \quad \text{Exact}$</td>
</tr>
<tr>
<td><strong>Turbulent diffusion</strong></td>
<td>$D_{r,ij} = -\frac{\partial}{\partial x_k} \left[ \rho \overline{u_i u_j} + p \left( \delta_{ij} + e_{ijk} u_j \right) \right] \quad \text{Modeled by GGD Daly and Harow [14]}$</td>
</tr>
<tr>
<td><strong>Simplification</strong></td>
<td>$D_{r,ij} = \frac{\partial}{\partial x_k} \left[ \mu \overline{u_i u_j} / \sigma_k \right] \quad \text{Scalar diffusivity by [15] to avoid numerical instabilities.}$</td>
</tr>
<tr>
<td><strong>$\mu_\epsilon$ and $C_\mu$</strong></td>
<td>$\rho C_\mu \frac{k^2}{\epsilon} \text{ and } 0.09 \quad \text{Same as } k-\epsilon \text{ models}$</td>
</tr>
<tr>
<td><strong>$\sigma_k$</strong></td>
<td>0.82 derived by [15] applying GGD to planar homogeneous shear flow \text{Different from standard and realizable } k-\epsilon \text{ models }=1</td>
</tr>
<tr>
<td><strong>Pressure-strain</strong></td>
<td>$\phi_{ij} = \phi_{ij,1} + \phi_{ij,2} + \phi_{ij,w} \quad \text{Gibson and Launder [11], Fu et al. [16], and Launder [17], [12]}$</td>
</tr>
<tr>
<td><strong>$\phi_{ij,1}$ Slow term</strong></td>
<td>$-C_1 \rho \frac{\epsilon}{k} \left[ \overline{u_i u_j} - \frac{2}{3} \delta_{ij} k \right] \quad \text{Return-to-isotropy term}$</td>
</tr>
<tr>
<td><strong>$\phi_{ij,2}$ Rapid term</strong></td>
<td>$-C_2 \left[ (P_{ij} + F_{ij} + G_{ij} - C_{ij}) - \frac{2}{3} \delta_{ij} (P + G - C) \right] \quad \text{Damps the normal stress perpendicular to the wall, while enhancing the stresses parallel to the wall.}$</td>
</tr>
<tr>
<td><strong>$\phi_{ij,w}$</strong></td>
<td>Responsible for the redistribution of normal stresses near the wall.</td>
</tr>
<tr>
<td><strong>LRN Modification</strong></td>
<td>With $C_1$, $C_2$ ... as functions of Re stress invariants and $\text{Re}_\tau = \left( \rho k^2 / \mu \epsilon \right)$ \text{Launer and Shima [18]}</td>
</tr>
<tr>
<td><strong>Buoyancy production</strong></td>
<td>$G_{ij} = \beta \frac{\mu_\epsilon}{\rho \text{Pr}<em>\tau} \left( g_i \frac{\partial T}{\partial x_j} + g_j \frac{\partial T}{\partial x_i} \right) \quad \text{Pr}</em>\tau = 0.85$</td>
</tr>
</tbody>
</table>
| **Modeling } k \text{**} | $k = \frac{1}{2} \overline{u_i u_i}$ \text{Transport equation is solved for } k \text{ to obtain B.C's for Re stresses} \text{When } k \text{ is needed for modeling specific term} \text{Identical to the equation used in standard } k-\epsilon \text{ model.} $
<table>
<thead>
<tr>
<th>Modeling $\varepsilon_y$</th>
<th>$\varepsilon_y = \frac{2}{3} \delta_y (\rho \varepsilon + Y_M)$, where $Y_M = 2 \rho \varepsilon M_i^2$</th>
<th>Additional “dilatation dissipation” by Sarkar [19] $M_i = \sqrt{k / a^2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Scalar $\varepsilon$</td>
<td>Model transport equation</td>
<td>Similar to that used in standard $k-\varepsilon$ model.</td>
</tr>
</tbody>
</table>

2.3 Turbulent Heat Transfer Models

The time averaging process of the energy equation results in extra flux terms $q_i = -\rho u_i \overline{T}$ which need to be modeled. Several heat transfer models exists in open literature. The SGDH (Simple Gradient Diffusion Hypothesis) is the simplest heat transfer model. It is based on similarities to the molecular heat transfer:

$$u_i \overline{T} = -\varepsilon \frac{\partial T}{\partial x_i} = -\frac{v_i}{\text{Pr}_t} \frac{\partial T}{\partial x_i}$$  \hspace{1cm} (2.14)

where $\text{Pr}_t$ is the turbulent Prandtl number.

GGDH (Generalized Gradient Diffusion Hypothesis) is a heat transfer model suitable when RSM is employed for the flow field. It is an adaptation of the Daly-Harlow diffusion model and relays on similarities (an extended Reynolds analogy) to the flow field. The model can incorporate unalignment effects in the relations for the heat flux vector and the temperature gradient:

$$-\overline{u_i T} = C_\theta u_i u_j \frac{k}{\varepsilon} \frac{\partial T}{\partial x_j}$$ \hspace{1cm} (2.15)

Launder [20] derived a scalar-flux transport equation similar to the RSM. The complexity of these models, and the number of tuned coefficients needed, made their use so limited.
In [21], [22], [23], [24] and [25] it was possible to derive a $k - \varepsilon$ model from the flux transport equation, similar to a two equation EVM.

Similar to EARSMs, [26], So and Sommer [27] and Wikstrom et al. [28] derived algebraic equation for the heat flux vector to construct a heat transfer model.

Thorough studies on rotational induced turbulence for the heat transfer are difficult to find in open literature. One-way to do that is to treat rotational induced turbulence in analogy to buoyancy-induced turbulence, see Launder [29].

In this dissertation, turbulent heat transfer is modeled using the concept of Reynolds’ analogy to turbulent momentum transfer. Thus the modeled energy equation is given as follows:

\[
\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i} \left[ u_i (\rho E + p) \right] = \frac{\partial}{\partial x_i} \left[ \left( k + \frac{c_p \mu_t}{\text{Pr}_t} \right) \frac{\partial T}{\partial x_i} + u_i (\tau_{ij})_{\text{eff}} \right] + S_h \tag{2.16}
\]

where $E$ is total energy, $(\tau_{ij})_{\text{eff}}$ is the deviatoric stress tensor, given by:

\[
(\tau_{ij})_{\text{eff}} = \mu_{\text{eff}} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \mu_{\text{eff}} \frac{\partial u_i}{\partial x_i} \delta_{ij} \tag{2.17}
\]

The viscous heating is represented by the term $(\tau_{ij})_{\text{eff}}$. $\text{Pr}_t$ is the turbulent Prandtl number with a default value of 0.85.
CHAPTER THREE: MODELING CONSIDERATIONS OF FLOW PHENOMENA IN ROTATING CHANNELS

The complex geometry, imposed Coriolis and centrifugal forces and the boundary conditions within the cooling channels inside rotors of electric generators and turbine blades complicate the flow and heat transfer fields. In this section these factors and how they influence the flow and heat transfer are discussed.

3.1 Cooling Channels Walls

Cooling channels walls have several effects on flow and turbulence. These effects are:

- No-slip condition, because of viscous effects.
- The blocking effect that makes the turbulence anisotropic by suppressing the fluctuations in the wall normal direction.
- The shearing mechanism in the flow increases the turbulent production.
- Wall reflection effect through reduction of stress components redistribution.
- Effect of surface roughness.

To account for the wall damping effect, Prandtl [30] proposed a reduction of the turbulent length scale as a function of the wall distance. Jones and Launder [31] and later Hanjalic and Launder [32] introduced LRN modified k-\(\varepsilon\) and LRN RSM turbulence model respectively to account for
viscous damping through a turbulent Reynolds number. Gibson and Launder [11] proposed a model for the wall reflection redistribution effect via additional terms in the pressure strain model. To ensure that the turbulent anisotropy close to walls could be reproduced, Durbin [33] modeled the viscous damping and the wall redistribution effects using elliptic relaxation equations in RSM. In recent paper Bredberg [34] employed a variety of damping functions and wall influenced terms to improve wall treatment in LRN modeling.

High Reynolds number (HRN) turbulence models deal with near-wall region using wall-functions that are based on the logarithmic law of the wall. As these functions are derived from abridged governing equations, their universality is limited. Many researchers have proposed a number of improvements, see [35].

The surface roughness effects are of considerable importance. When the walls are smooth, the shear stress at the surface is transmitted to the flow via a viscous sublayer, where the velocity in this sublayer varies linearly, as in laminar Couette flow, \( U = u_r^2 y / v \). This implies that the thickness of the sublayer is on the order of \( \delta_{\text{lam}} = (\text{const}) v / u_r^2 \). The constant is the value of the velocity where it ceases to behave linearly. This implies a law given by: \( U^* = \frac{1}{\kappa} \ln(y^*) + B \), with \( B \) a constant that accounts for conditions at the boundary to add a uniform velocity to the entire flow with no change in its internal structure. For a rough surface with roughness height larger than \( \delta_{\text{lam}} \), the stress is transmitted by pressure forces in the wakes of the roughness elements, rather than by viscosity. The form of the profile given in the log layer is then: \( U^* = \frac{1}{\kappa} \ln\left(\frac{y}{k_s}\right) + 8.5 \), where \( k_s \) is Nikuradse’s equivalent roughness height, and is related to the
roughness geometry at the boundary. For boundary layers that are considered fully rough
\((k^+ = ku_r / \nu > 15)\), estimation for the friction coefficient for zero pressure boundary layers may
be: \(C_f = \left[ 2.87 + 1.58 \log \left( \frac{x}{k_s} \right) \right]^{-2.5} \). Numerically, the wall node must be placed at some distance
above the equivalent roughness \(k_s\). This follows from the physical argument that the flow cannot
exist inside the wall, whose edges effectively protrude a distance \(k_s\) into the flow.

3.1.1 Modeling Wall Roughness Effects

Wall roughness affects drag (resistance) and heat and mass transfer on the walls. Experiments in
roughened pipes and channels indicate that the mean velocity distribution near rough walls, when
plotted in the usual semi-logarithmic scale, has the same slope \((1/k)\) but a different intercept
(additive constant \(B\) in the log-law). Thus, the law-of-the-wall for mean velocity modified for
roughness has the form

\[
\frac{u_p u^*}{\tau / \rho} = \frac{1}{k} \ln \left( E \frac{\rho u^* y_p}{\mu} \right) - \Delta B
\]

(3.1)

where \(u^* = C_{\mu}^{1/4} k^{1/2}\) and

\[
\Delta B = \frac{1}{k} \ln f_r
\]

(3.2)

where \(f_r\) is a roughness function that quantifies the shift of the intercept due to roughness
effects. \(\Delta B\) depends on the type (uniform sand, rivets, threads, ribs, mesh-wire, etc.) and size of
the roughness. For a sand-grain roughness and similar types of uniform roughness elements, \(\Delta B\)
has been found to be well correlated with the nondimensional roughness height,
\[ K_s^+ = \frac{\rho K_s u^*}{\mu}, \] where \( K_s \) is the physical roughness height. Analyses of experimental data show that the roughness function is not a single function of \( K_s^+ \), but takes different forms depending on the \( K_s^+ \) value. It has been observed that there are three distinct regimes:

- Hydrodynamically smooth \((K_s^+ \leq 2.25)\)
- Transitional \((2.25 < K_s^+ \leq 90)\)
- Fully rough \((K_s^+ > 90)\)

In FLUENT, the whole roughness regime is subdivided into the three regimes, and the formulas proposed by Cebeci and Bradshaw [36] based on Nikuradse's data are adopted to compute \( \Delta B \) for each regime.

In the solver, given the roughness parameters, \( \Delta B \) the modified law-of-the-wall in Equation 3.1 is then used to evaluate the shear stress at the wall and other wall functions for the mean temperature and turbulent quantities.

### 3.1.2 Near Wall Treatment

In light of this dissertation both wall functions by Launder and Spalding [37] and enhanced near wall treatment approaches, will be used and compared for near wall treatment. Table 3.1 and Table 3.2 provide a summary of the standard wall functions and enhanced near wall treatment approaches, respectively.
The standard wall functions are semi-empirical formulas used to bridge the viscous sublayer and buffer layer near the wall, while in the enhanced wall treatment approach the Viscosity-affected region is resolved all the way to the wall, including the viscous sublayer.
### Table 3.1 Near-wall Treatment using Standard Wall Functions

<table>
<thead>
<tr>
<th>Description</th>
<th>Semi-empirical formulas are used to bridge the Viscous sublayer and buffer layer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advantages</td>
<td>Saves computational resources, economical, robust, and reasonably accurate.</td>
</tr>
</tbody>
</table>
| Disadvantages: Limitations | - Inadequate where the LRN effects are pervasive  
- Constant-shear assumption  
- Local equilibrium hypotheses |

#### Momentum

$$ U^* = \frac{1}{\kappa} \ln \left( E y^* \right) $$

Low of the wall

#### Log-law

$$ U^* = \frac{U_p C_{\mu}^{1/4} k_p^{1/2}}{\tau_w / \rho} $$

$$ y^* = \frac{\rho C_{\mu}^{1/4} k_p^{1/2} y_p}{\mu} > 30 - 60 $$

#### Laminar stress-strain relationship

$$ U^* = y^* $$ when $ y^* < 11.225 $

#### Energy

$$ T^* = (T_w - T_p) \rho C_p C_{\mu}^{1/4} k_p^{1/2} / q^* $$

#### Thermal sublayer thickness, $ y_T^* $$

Computed as the $ y^* $ value at which the linear law and the logarithmic law intersect, given the molecular Prandtl number of the fluid

#### Turbulence

$$ G_k \approx \tau_w \frac{\partial U}{\partial y} = \tau_w \frac{\tau_w}{\kappa \rho C_{\mu}^{1/4} k_p^{1/2} y_p} $$

$$ \varepsilon_p = \frac{C_{\mu}^{3/4} k_p^{3/2}}{\kappa y_p} $$

#### Not Recommended

- Pervasive low-Reynolds-number (e.g., flow through a small gap or highly viscous, low-velocity fluid flow)  
- Massive transpiration through the wall (blowing/suction)  
- Severe pressure gradients leading to boundary layer separations  
- Strong body forces (e.g., flow near rotating disks, buoyancy-driven flows)  
- High three-dimensionality in the near-wall region (e.g., Ekman spiral flow, strongly skewed 3D boundary layers)

#### Near-Wall mesh consideration

- $ y^* \approx 30 $  
- Avoid using excessive stretching in the normal direction.  
- Have at least a few cells inside the boundary layer.
Table 3. 2 Enhanced Near Wall Treatment using Combined Two Layer Model with Enhanced Wall Functions

<table>
<thead>
<tr>
<th>Description</th>
<th>To have a near-wall modeling approach that will possess the accuracy of the standard two-layer approach for fine near-wall meshes and that will not significantly reduce accuracy for wall-function meshes. Viscosity-affected region is resolved all the way to the wall, including the viscous sublayer</th>
</tr>
</thead>
<tbody>
<tr>
<td>1- Two-Layer</td>
<td>Used to specify both $\varepsilon$ and the turbulent viscosity in the near-wall cells, the whole domain is subdivided into a viscosity-affected region and a fully-turbulent region. The demarcation of the two regions is determined by a wall-distance-based, turbulent $Re_y$</td>
</tr>
<tr>
<td></td>
<td>$Re_y = \frac{\rho y \sqrt{k}}{\mu}$, $y$ = normal distance from the wall at the cell centers</td>
</tr>
<tr>
<td></td>
<td>In the fully turbulent region ($Re_y &gt; Re_y^* ; Re_y^* = 200$), $k - \varepsilon$ or RSM are employed</td>
</tr>
<tr>
<td></td>
<td>In the viscosity-affected near-wall region ($Re_y &lt; Re_y^*$), one-equation model of Wolfstein [38] is employed</td>
</tr>
<tr>
<td>In one-equation model</td>
<td>Momentum and $k$ equations remain the same</td>
</tr>
<tr>
<td></td>
<td>$\mu_{t,2layer} = \rho C_\mu \mu \sqrt{k}$, $\ell_\mu = y C_1 \left(1 - e^{-Re_y/A}\right)$, [39]</td>
</tr>
<tr>
<td>Enhanced wall treatment</td>
<td>$\mu_{t,enh} = \lambda_c \mu_t + (1 - \lambda_c) \mu_{t,2layer}$, where $\mu_t$ is the HRN definition [40]</td>
</tr>
<tr>
<td>Blending function</td>
<td>$\lambda_c = \frac{1}{2} \left[1 + \tanh \left(\frac{Re_y - Re_y^*}{A}\right)\right]$</td>
</tr>
<tr>
<td></td>
<td>= 1 far from walls and = 0 near walls</td>
</tr>
<tr>
<td></td>
<td>Prevents solution convergence from being impeded when the $k - \varepsilon$ solution in the outer layer does not match with the two-layer formulation</td>
</tr>
<tr>
<td>Enhanced wall functions</td>
<td>Blending linear (laminar) and logarithmic (turbulent) laws-of-the-wall. This approach allows the fully turbulent law to be easily modified and extended to take into account other effects such as pressure gradients or variable properties</td>
</tr>
<tr>
<td></td>
<td>$u^+ = e^T u_{lam}^+ + e^T u_{turb}^+$, Kader [41]</td>
</tr>
<tr>
<td></td>
<td>$\Gamma = \frac{a(y^+)^4}{1 + by^+}$ blending function</td>
</tr>
<tr>
<td></td>
<td>$\frac{du^+}{dy^+} = e^T \frac{du_{lam}^+}{dy^+} + e^T \frac{du_{turb}^+}{dy^+}$</td>
</tr>
<tr>
<td></td>
<td>This formula guarantees the correct asymptotic behavior for large and small $y^+$ and reasonable representation of velocity profiles in the cases where $y^+$ falls inside the wall buffer region ($3 &lt; y^+ &lt; 10$).</td>
</tr>
<tr>
<td>Enhanced thermal wall functions</td>
<td>$T^+ = e^T T_{lam}^+ + e^T T_{turb}^+$ where $\Gamma = \frac{a(Pr y^+)^4}{1 + b(Pr y^+)}$</td>
</tr>
</tbody>
</table>

35
B.C for \( k \)

\[
\frac{\partial k}{\partial n} = 0 \quad \text{Same as for standard wall functions}
\]

\( G_k \)

Computed using the velocity gradients that are consistent with the enhanced law-of-the-wall \( u^+ \) and \( \frac{du^+}{dy^+} \)

Near-Wall mesh consideration

\( y^+ = 1 \) However, \( y^+ < 4-5 \) is acceptable

- Have at least 10 cells within the viscosity-affected near-wall region (Re_\( y \) < 200) to resolve the mean velocity and turbulent quantities.

3.2 Effect of Ribs

Ribs are introduced inside the cooling ducts to enhance heat transfer. Usually ribs are placed along leading and trailing walls to increase wall roughness, which increases both the heat transfer and the friction of the duct, and consequently increases the pressure drop several times higher than the case of smooth ducts.

The rib size \( e \) and blockage ratio \( e/D_h \) has a strong effect on the pressure drop and the heat transfer. Ribs usually arranged orthogonal to the flow to the stream wise direction. Ribs, while usually made of square cross section other shapes and skewed ribs, Abdon [42], Han et al. [43], [44], [45], [46] proved to have better heat transfer. Webb et al. [47] investigated the importance of the pitch \( P \), the distance between two successive ribs and rib-width, which does not affect the heat transfer much.

For rib roughened rotating channels with relatively low rotation numbers the turbulence level and hence the heat transfer increase by a factor of 2-4 compared to smooth ducts because recirculating zones, shear, mixing, and impinging of the flow are formed around the ribs. Vogal and Eaton [48] studied the large re-circulating zone downstream the ribs and derived a relation
between Nusselt number and Reynolds number as: $Nu \sim Re^{0.6}$. Further more it was concluded that the Reynolds analogy that connects between skin friction and heat transfer, fails in separated regions.

![Flow](image)

**Figure 3.1** Velocity Contours and Velocity Vectors around a Rib Showing Re-Circulating bubble. RSM Predictions with Enhanced Wall Treatment

In rib-roughened channels with repeated ribs the flow pattern around ribs depends on the shape and size of the rib, and the pitch P. Figure 3.1 shows the velocity contours and vectors predictions using RSM with enhanced wall treatment for the case studied experimentally by Rau et al. [49], where they used in-line ribs with square cross section with blockage ratio, $c/Dh$ of 10% and $P/e$ of 9.5. In addition to the separation bubbles around the ribs, there is an additional bubble appears on the rib top.
Performing experimental investigations on the flow pattern within two successive ribs for channel with rectangular periodic ribs, Webb et al. [47] concluded that when $P/e > 15$, a long re-circulating zone (about 6 to 8 rib-height) formed behind the rib. When the pitch is smaller, the upstream and the downstream re-circulating zones start to interact until $P/e \approx 8$. For $P/e < 8$, a single large re-circulating bubble is formed and the flow dose not reattach. It has been shown that the best heat transfer rate can be achieved when the flow re-attaches between the ribs, without redeveloping before separating due to blockage effect of the next rib. Liou and Hwang [50] found for the rib-roughened flow that the Nusselt number and turbulent kinetic energy are well correlated in the separated region behind the rib. The peak of Nusselt number was located around one rib-height upstream of the re-attachment point. Using k-ε and k-ω models in rib-roughened 2D channels, Bredberg et al. [51] obtained corresponding results. They showed that the Nusselt number differed by nearly a factor of two even though similar levels of $k$ were predicted. It is argued that the representation of the length-scale is vital for the correct assessment of heat transfer in separated flows using two-equation models. This is also the argument behind the length-scale correction of Yap [52]. Panigrahi and Acharya [53] gave a detailed study on turbulent mechanism around ribs using an octant analysis. Schere and Witting [54] found that the discretization scheme might be a source of error in location of maximum heat transfer behind BFS. Bredberg [51], using LRN k-ε and k-ω turbulence models, showed that the use of wall functions or even the choice of turbulence model might be the reason behind inaccurately predicted re-attachment point.
Han et al. [55] showed that ribs with an angle of 45 deg to the streamwise direction increase the heat transfer rate by 25% compared to normal ribs arrangement for the same pressure drop. The reason might be that skewed ribs produce a secondary motion that flows transversally along the rib due to induced cross-stream pressure gradient. In orthogonal ribs similar pattern is not found, a much weaker secondary flow with downward motion (towards the rib) along the centerline and upward along the side-wall in a circulating fashion is present. Rau et al. [49] and Liou et al. [56] attributed this motion to both the difference in the static pressure, and turbulent generating processes (which contributes to Prandtl [57] secondary flow of the second kind). Two-equation eddy-viscosity models, such as the $k-\varepsilon$ turbulence model, are unable to reproduce turbulence generated secondary flows (due to their isotropic assumption of the Reynolds stresses), but still capture the vertical motion via the pressure difference.

As a result of increasing the rib height, angled ribs, reduced pitch etc., the flow becomes more three-dimensional and the more uncertain the correlations that are valid for a wide range of configurations. In design situations with such uncertain correlations it is worth to do optimization using numerical simulations and it is necessary to have detailed experimental data, for example the experimental data by Drain and Martin [58], where they performed a laser Doppler velocimetry (LDV) on several rib-configurations.

The effect of Reynolds number for stationary rib-roughened channels was studied by Bredberg [51], [59] using three types of turbulence models and showed that turbulence models that agree well with a low Reynolds numbered case, over predicted the heat transfer for a high Reynolds numbered case. Kukreja et al. [60] investigated the effect of span wise distribution on Sherwood
number of differently angled ribs, and found that the vortical cells produced by the angled ribs produce a very complex pattern in the local mass transfer which is impossible to correlate to any empirical function. Even for orthogonal ribs, a substantial variation of the heat transfer in the spanwise direction is evident.

The results from these and similar experiments show the necessity to perform three-dimensional computations. The detailed study by Rau et al. [49] included both flow field and heat transfer measurements and it is used in this dissertation for turbulence model evaluation for the stationary rib-roughened case.

3. 3 Secondary Flows

According to Prandtl [57] there are two types of secondary flows:

- Secondary flows generated by inviscid effects – First kind secondary flows.
- Secondary flows generated by Reynolds stresses – Second kind secondary flows.

The first kind secondary flow is generated when the spanwise and either of the vorticity components generated. The main re-circulating bubble in ribbed channel flow is not a secondary flow, as the ribs only generate spanwise vorticity, through an increased shearing. Second type secondary flows generated by Reynolds stresses (turbulence induced) develop in corners of the duct; where the cross-stream gradients of the Reynolds stresses generate weak streamwise vorticity. Because these secondary flows are driven by gradients of Reynolds stresses they cannot be captured using an isotropic eddy-viscosity models. Figure 3. 2 shows the two types of secondary flows where the first type secondary flow that is generated by rotational induced
Coriolis force appears in the lower half of the square duct, while the second type secondary flow due to turbulence can be seen in the top corners of the duct.

A review of modeling achievements compared to experimental results has been performed by Demuren and Rodi [61]. Huser et al. [62], based on DNS-data, concluded that no turbulence model could accurately predict the turbulence characteristic of Prandt’s secondary flow generated by Reynolds stresses although the basic secondary motion can be predicted using non-linear models. This has been confirmed by a recent computation by Petterson and Andersson [63] using an RSM with an elliptic relaxation and the non-linear SSG pressure-strain model. However, the influence of the Reynolds stress induced secondary flow is less important in complex and disturbed flows because they have a weak structure (2 – 4%) of the bulk velocity.
Bredberg [64] showed that for rotating rib-roughened channels, where the flow is characterized by large-scale separation and rotational induced secondary flows, there is only a minor modification by adding non-linear terms (EARSM) to the eddy viscosity turbulence model.

First kind secondary flows governed by inviscid effects are simpler to predict and also more influential on the flow structure. This kind of secondary flows is generated either by a pressure gradient (geometrical conditions, e.g. Ribs) or an imposed body-force (Coriolis force due to rotation or buoyancy) in the cross-stream plane. The void behind the rib in rib roughened channels generates a downward motion within the re-circulating region, which accompanied by the no-slip condition will generate a circular motion with two opposite rotating cells as shown by Hirota et al. [65]. Kukreja et al. [60] found that in a V-configuration rib arrangement the highest pressure is found in the center of the duct, at any given streamwise location. Such an arrangement enhances the strength of these cells and the re-distribution of the heat transfer pattern.

3.4 U-Bends – Effect of curvature

The internal cooling channels of gas turbine blades are usually consist of two or more straight passages with U-bends. As the fluid reaches the bend in channels with zero rotation number, a centripetal acceleration is generated, which is balanced by an opposing pressure gradient. For bends with high radios, the pressure gradient varies almost linearly from the inner to the outer radii. The high momentum fluid in the duct center tends to drift outwards. Continuity requires that the outward motion in the center of the duct be balanced by a reverse flow along the walls,
where the centrifugal force is less because the streamwise velocity is lower. This flow behavior generates a circular motion in the cross-stream plane. At the exit of the bend two opposing rotating cell, with vorticity in the streamwise direction will appear, see Figure 3. The Dean number [66], $\text{De}$ is the parameter that measures the curvature effect relative to viscous effect ($\text{De} = \text{Re}(D_h/R)^{1/2}$), where $\text{Re}$ is the Reynolds number and $R$ is the radius of the curvature. Dean number gives a measure of the degree of stability. The flow is considered unstable when exceeding the critical Dean number according to Rayleigh’s criteria. For the flow in a bend the convex side (inner) is stable, while the concave side (outer) is unstable. The fluid viscosity affects the stability of the flow through bend and hence it is important for turbulence modeling.

![Figure 3. 3 Secondary Flows at the Center (Left) and Exit (Right) of Square Cross Section U-bend - Stationary Case, Re=25,000, DR=0.13, RSM Predictions.](image)

Many researchers proposed turbulence modeling modifications due to curvature effect. Prandtl [67] proposed to add correction to the mixing length, based on a local dimensionless curvature parameter:

$$ \frac{U/R}{du/dy} $$

(3.3)
To modify the turbulent length scale, Bradshaw [68] used Richardson number. Note that this modification is not applicable in the standard two-equations models (k-ε, k-ω) because the turbulent length scale is not given explicitly. Launder et al. [69] introduced a Richardson number modification in the ε-equation. Since streamline curvature makes redistribution among the Reynolds stresses, Wilcox and Chambers [70] argued that the modification should be applied to the turbulent kinetic energy. Lunder [12] showed that the isotropic EVMs have a limited capability to accurately predict the complex secondary flows in a bend. This is clear from the fact that under unstable conditions (concave side) the wall normal component is amplified and the streamwise component is reduced and for the convex side the opposite happens. Bredberg [64] questioned the strategy of Wilcox and Chambers for achieving improved predictions, through only redistribution among the stress components because the level of turbulence energy is different on the concave and convex sides of the bend. Instead he used Richardson number modified ε-equation and achieved more realistic profiles for the turbulent kinetic energy. Hellsten [71] and Shih et al. in more recent studies have focused on the ε-equation. Luo and Lakshminarayana [72] showed that even RSM benefits from modification to the length scale for channels with strong curvature.

3.5 Rotational Effects

The additional body forces terms, due to rotation, in the NS equation are the centrifugal and Coriolis:

\[ f_{i}^{ce} = -\rho \varepsilon_{ikm} \Omega_{j} \Omega_{j} x_{m} \]  \hspace{1cm} (3.4)

\[ f_{i}^{co} = -2\rho \varepsilon_{ijk} \Omega_{j} U_{k} \]  \hspace{1cm} (3.5)
Figure 1. 3 in chapter one shows the rotation and curvature induced secondary flows in a U-bend rotating around -x-axis. The Coriolis force $f^{co} = \pm 2|\Omega||U|$ is parallel to the y-axis, and generates a secondary flow in the cross-stream plane, y-x. The curvature effect produces a similar secondary motion within the bend and in the second passage. When the main Coriolis component is perpendicular to the surface the fluid will either be forced back, or driven away from the surface, dependent on the direction of the Coriolis force. If the rotational induced force is directed into the surface the flow becomes unstable. A stabilized boundary layer suppresses turbulence, and can under strong rotation even laminarize. In square duct the stress-induced corner vortices are significantly reduced for increasing rotational number on the stable side. Thus a higher order turbulence model is needed for accurate numerical simulation of rotating flows, although the main vortices, governed by the inviscid Coriolis force, are captured using even the simplest flow model.

Pallares and Davidson [73] studied the interaction of Prandtl's secondary flow of the second kind (stress induced) with those of the first kind (rotational-induced) using LES model and showed that for increasing rotational numbers, the corner vortices are suppressed and that the cross-stream secondary motion is mainly governed by the Coriolis force.
CHAPTER FOUR: LITERATURE REVIEW

4. 1 Work with Reference to Orthogonal Mode Rotation

There is an increasing demand to improve the cooling of gas turbine blades through experiments and computational methods. When performing experiments, in addition to the high cost, it is very difficult to achieve conditions, which enable measured data to be dependent on few parameters. In many experiments the level of uncertainty of measurements alters the data. In numerical simulations it is possible to insure accurate values for all variables and boundary conditions. It is thus important to perform both measurements and predictions to study the flow field and heat transfer in such complex situations.

In this section experimental and numerical work of heated rotating channels with smooth and with rib-roughened walls in orthogonal and parallel mode rotation are considered. Special consideration is given to the effect of rotation number and density ratio.
4.1 Channels with Smooth Walls

Many research groups have performed studies on channels with smooth walls both experimentally and numerically. This effort, although has been performed mostly for relatively low rotation numbers and density ratio, is to be addressed next.

4.1.1 Effect of Rotation Number in Smooth Wall Channels - Experimental Work

Rotation number, by definition is a relative measure of Coriolis force to the bulk flow inertial force. A higher rotation number signifies Coriolis and Buoyancy effects on flow and heat transfer. Wagner et al., [74] found that increasing the rotation number up to 0.475 and density ratio of 0.13 causes the Nusselt number in the first pass of the trailing side to increase by a factor of 4 compared to stationary case; however, the heat transfer decreases rapidly in the second pass to the same values as for stationary case. In the leading side the increase by a factor of 1.5 was observed on the second pass and a decrease of heat transfer in the first pass. In the U-turn, the heat transfer increased by a factor of more than 4.7 and 4.4 in the trailing and leading sides respectively. The decrease in Nusselt number on the second pass of the trailing side and the first pass of the leading side is due to the decrease in the axial flow and the stabilization of the near-wall flow on the leading side.

Many researchers, Moore, 1967; Hart, 1971; Wagner and Velkoff, 1972; Johnson et al., 1972; Rothe and Johnson, 1979 (see Wagner et al. [75]) studied the effect of rotation (with relatively low rotation numbers, less than 0.24) in an unheated circular and rectangular passages and reported secondary flows and stability aspects of the flow.
4.1.1 Effect of Rotation Number in Smooth Wall Channels - Numerical Simulations

All numerical investigations found in open literature were performed for relatively low rotation numbers and density ratios. The main objectives of these investigations were to compare and verify the capabilities of different turbulence models and to study the heat transfer and flow for the same range of rotation numbers and density ratios studied previously experimentally. Majumdar et al. [76] used k-ε model to compare predictions with measurements, where the heat transfer predictions were not so successful. Tekriwal [77] used extended k-ε model and showed that the predictions depend on grid distribution and y+. Parkash and Zerkle [78] employed high Reynolds number k-ε model with wall functions approach. They concluded that the k-ε model heat transfer predictions were inaccurate and more refined model is needed. Sathyamurthy et al. [79] used k-ε model with wall function treatment to predict flow and heat transfer in square rotating duct with U-turn. They concluded that more advanced model (RSM) with low Reynolds number is needed to predict such complex flow. Performing computations on three pass channel with two U-turns, McGrath and Tse [80] concluded that the two-layer k-ε model showed improvement over the wall functions treatment. Applying a modified k-ε model, Dutta et al. [81] showed satisfactory predictions on the radial outward flow. Stephens et al. [82] used k-ω model to simulate the smooth square duct with U-turn. The k-ω model showed good heat transfer results agreement with those measured by Wagner [74] except at the leading surface of the first pass where an overestimation was predicted. Studying duct flow with curvature and rotation, Iacovides et al. [83] used high Re k-ε model, high Re algebraic second-moment (ASM) closure, low-Re ASM with the dissipation rate obtained algebraically and low-Re RSM with ε transport equation solved over the entire domain. They concluded that the complex flow downstream the
bend is not reproduced well by the low-Re RSM for the rotating case. Bohnoff et al. [1] implemented RSM model using FLUENT commercial code with standard wall functions. The average heat transfer predictions were close to Wagner et al. [74] except a slight overestimation in the second pass. Chen et al., [84] simulated the case studied previously by Wagner et al. [74]. They used a near wall second-moment closure model and compared to a two-layer k-e model, which performed worst. The comparison with experimental data clearly demonstrated that the secondary flows in rotating two-pass channels have been strongly influenced by the Reynolds stress anisotropy resulting from the Coriolis and centrifugal buoyancy forces as well as the U-turn wall curvature.

4.1.1 3 Effect of Density Ratio in Smooth Wall Channels - Experimental Work

Wagner et al. [74] found that increasing the density ratio from 0.07 to 0.22, i.e., rotational buoyancy, with Reynolds number of 25,000, rotation number of 0.24, and rotating radius of 49 hydraulic diameter, causes a 50% increase in heat transfer ratio in the first pass trailing surface and a 100% increase in the first pass leading surface. In the second pass the heat transfer coefficient increases by increasing density ratio. In the U-turn the heat transfer increased by more than 30% on the trailing and leading surfaces.

Many researchers have studied the effect of density ratio without rotation; see [75] for more references, in vertical stationary smooth ducts. Their results were used later in studying flow and heat transfer in rotating machinery.
4.1.1 Combined Effects of Rotation Number and Density Ratio - Experimental Work

Wagner et al., [74] showed that there are no effects of increasing density ratio on Nusselt number ratio for stationary case, because the gravitational buoyancy is negligible and the rotational buoyancy is zero at zero rotation number. Increasing the rotation number to 0.3 and density ratio to 0.22 causes the Nusselt number to increase on the trailing surface of the first pass compared to lower density ratios. In the leading surface of the first pass heat transfer decreases by increasing rotation number and then increases again, especially with higher density ratios. Wagner et al. [85] attributed the heat transfer behavior on the leading surface to the combination of buoyancy forces and the stabilization of the near wall flow and to the Coriolis-driven secondary flow cells at larger rotation numbers. Heat transfer ratio in the second pass is relatively unaffected by increasing density ratio and rotation number. The thermal boundary layers at the exit of the U-turn, i.e. near the inlet of the second pass, are thin. The turn dominated secondary flows and the rotational effects on heat transfer become more prominent with increasing the axial distance from the U-turn.

Heat transfer in smooth walls rotating channels has been studied by Mori et al. (1971), Johnson (1978), Lokai and Gunchenko (1979), Morris and Ayhan (1979), Morris (1981) Isakov and Trushin (1983), and Clifford (1985), (see Han et al. [86]). Their investigations have been conducted for low rotation numbers, less than 0.2, and low-density ratio, less than 0.2. However, their results were inconsistent. The test conditions, measurement techniques and models might be the reason for the inconsistency.
4.1.1 Combined Effects of Rotation Number and Density Ratio- Numerical Simulations:

The predictions by Parkash and Zerkle, [78] and Dutta et al. [81] showed that the increase in heat transfer coefficient on the leading surface is related to buoyancy driven flow separation. Dutta et al. [87], using the proper inlet conditions of the experiment, obtained satisfactory heat transfer results using $k-\varepsilon$ model. Bonhoff et al. [88] and Iacovides et al. [89] studied the same rotating smooth U-bend case using higher order turbulence models (RSM and ASM respectively). Similar to previous studies by the UMIST-group (Launder, Iacovides) the latter studied employed one equation model in the near wall region, while [88] used standard logarithmic wall function in a commercial code. For a lower Reynolds number case, $Re = 25 000$, Stephens and Shih [82] were able to use a LRN $k-\omega$ turbulence model on a $1.1x10^6$ mesh for their compressible simulation. The result was in good agreement with the measured data. Bredberg [90] estimated that the required number of nodes on a well resolved mesh for $k-\omega$ amounted to $2 \times 10^6$. Through a number of simplifications many researchers have been able to reduce the amount of nodes to $1.9 \times 10^6$.

4.1.2 Rib Roughened-Wall Coolant Passage

Ribs are introduced on leading and trailing surfaces inside internal cooling channels to disturb the boundary layer and enhance the heat transfer by causing flow separation and increasing turbulence in boundary layer. The effect of rotation in ribbed channels is different from that in smooth channels since rotational forces are coupled with local velocity and temperature distributions. Ribs usually placed with 90 deg, 60 deg or 45 deg angles to the bulk flow.
Many researchers have studied rib roughened channels experimentally: Webb et al. [47] investigated rib-roughened circular tubes. Han et al. [55], [44], [43], [45], [46] investigated the effect of different rib cross-sectional shapes, skewed ribs and channel aspect ratio in order to enhance the heat transfer performance (also the friction coefficient/pressure drop). It was difficult to incorporate all the results in correlation or to curve-fit the data. Taslim [91] performed heat transfer coefficient measurements on 45-degree round corner ribs. The effect of a variety of rib configuration on heat transfer could be found in the van Karman Lecture series recently given by Taslim [92]. Rotating channels with rib-roughened walls were studied by Morris and Farhadi [93]. U-bends with and without rotation including curvature effect with roughened walls were studied also, (see [90] for more references). Experiments that simulate conditions in turbine blade need to include all effects (rotating, rib-roughened U-bends). Most experiments only measure heat transfer. Rathjen et al. [94] performed experimental and numerical heat/mass transfer investigations in rotating (Ro=0 and 0.1) two-pass channel with staggered 45 deg ribs. However Liou et al. [95] also gave an accurate flow field data using LDV. Uneven wall heat flux effects and review on experimental investigations could be found in [96] and [97]. Han et al. [86] provided both experimental and numerical review. Shih and Sultanian [98], gave a review on computations of internal and film cooling in gas turbines. However the following literature survey is only restricted to rotating channels.

4.1.2.1 Channels with 90 Ribs- Experimental Work

The 90 orthogonal ribs, unlike skewed ribs, do not create secondary flow of their own. An early experiment by Rothe and Johnson [99] showed the effect of rotation on the location of flow
reattachment after a backward facing step, which helped understanding the flow in ribbed roughened channels.

Wagner et al., [85] investigated orthogonal ribbed channels. The ribs were of circular cross section with fillets and staggered arrangement. For zero rotation number there is no effect of density ratio on heat transfer. In the trailing surface of the first pass increasing rotation number causes local increase in Nusselt number by 75 % compared to stationary case. For high pressure surfaces of the first pass the heat transfer increases rapidly with increasing either the rotation number or density ratio. In the second pass leading surface the heat transfer increases by 30 to 35%. In the low pressure surfaces (leading of the first pass and trailing of the second pass) the heat transfer from leading first pass decreases for low rotation numbers (Ro<0.25) and then, depending on density ratio, increases with increasing rotation number for larger values of density ratio.

Similar to the results obtained for smooth channels, Wagner et al. [85] showed that the heat transfer from the first pass trailing surface increase with density ratio. In the second pass trailing, the effects of density ratio are larger than in the first pass leading. Wagner et al. [85] attributed this to uneven heat transfer coefficient distribution on the low pressure surfaces, the combined effect of buoyancy forces and the stabilization of the near wall flow for low rotation numbers, the coriolis-driven secondary flow, and the increases in flow reattachment lengths downstream of ribs for higher rotation numbers. The thermal boundary layer near the inlet of the second pass is thin because of strong turbulence mixing and secondary flow effects. With increasing axial distance the effect of U-turn decreases and the effect of Coriolis-driven secondary cross-flow increased.
Taslim et al. [100] and Clifford [101] studied the effect of rotation on heat transfer in rib-roughened ducts for relatively low rotation numbers and density ratios. The results by different investigators showed clear inconsistency, which was attributed to the different between measurement techniques, models and test conditions. Using laser Doppler anemometry and wall pressure measurements, Iacovides et al. [102] compared the results of square U-bend staggered ribbed channel with smooth channel results for \( \text{Ro} = 0.2 \). They showed that the separation bubble after the turn is smaller in the ribbed channel than in the smooth one and found higher overall turbulence level and formation of a large separation bubble along the outer wall as the flow encounters the first outer-wall rib, after the bend exit. For the low rotation number considered in that study, the turn effect was stronger than the rotational effects, which resulted in minor influence of the rotational effects immediately downstream of the turn. The mean velocity distribution plots showed that the turn effects are stronger than the rotational effects in the downstream of the turn, while upstream of the turn the rotation increases the turbulence intensity.

In more recent experimental work, Liou et al. [95] performed LDV and transient thermochromic liquid measurements on square channel with U-turn and with square cross section in line 90-deg ribs arranged on leading and trailing surfaces. \( \text{Ro} \) was varied from 0 to 0.2 with \( \text{Re} \) fixed at 10,000. Their results showed that the rotation increased the streamwise velocity and the turbulence intensities, which leads to heat transfer enhancement. They also derived simple linear correlations between regional average Nusselt number and rotation number.
4.1.2. 2 Channels with 90 Ribs-Numerical Simulations

Parkash and Zerkle [103] performed a computation of rib-roughened rotating duct with cyclic streamwise boundary conditions, to reduce computational costs, and with neglected buoyancy-centrifugal effect. They used k-ε model with either wall function or a zonal approach for wall treatment. They suggested that a low Re RSM model is necessary to capture anisotropy turbulence effect. They showed that the flow separates and reattaches in the ribbed channel. It was observed that the rotational effects resulted in secondary flows and that the overall enhancement of heat transfer coefficients by rotation is less in a ribbed channel than in a smooth channel. Iacovides [104], [102] studied velocity and heat transfer in rotating channels with orthogonal staggered ribs using low Re Differential stress model (DSM) and effective viscosity model (EVM) with DSM performed better.

Liou et al. [56] performed both heat transfer measurements and numerical simulations using a k-ε algebraic stress turbulence model. The case studied was the one previously measured by Drain and Martin [58] using LDV. The results from Liou et al. [56] were used in the 7:th ERCOFTAC Workshop to evaluate CFD-codes and turbulence models. Significantly different heat transfer results were presented at the workshop. The extension using an EARSM formulation of the k-ω turbulence model did not improve the results much. Manceau et al. [105] applied the k-ε-ν²-f turbulence model with good results for the floor between the ribs. The heat transfer around the ribs was however not accurately captured although different boundary conditions were used.

Another test case in the workshop was a 3D rib-roughened channel with ribs placed on two opposite walls in staggered arrangement. The mean quantities were accurately predicted, however heat transfer was under predicted using the zonal EVM. The LRN k-ε model
necessitates the inclusion of a Yap-correction to yield acceptable heat transfer rates. Similar conclusions, for the same measurements data were drawn by Bredberg [59], when comparing a zonal $k-\varepsilon$ with a $k-\omega$ and EARSMs.

Iacovides and Raisee [106] applied EVM and second moment turbulence models to predict the flow and heat transfer through staggered ribbed–roughened passages with U-tern and with square ribs. They concluded that second moment closures are necessary in order to correctly reproduce the regions of flow separation, low –Re turbulent models are necessary for heat transfer computations and that a low-Re differential stress closure yields thermal predictions that are superior to those of the low-Re EVM model. Introducing a differential version of the Yap length scale correction term, independent of the wall distance, to the dissipation rate equation of the low-Re $k-\varepsilon$ model, they found improvement of heat transfer predictions.

4.1.3 Effect of Curvature

4.1.3.1 Effect of Curvature-Experimental Work

Experimental investigations on rotating U-channels have been performed by few researchers compared to stationary case. Wagner et al. [75], [74] performed heat transfer measurements in smooth wall channels with U-bend. Johnson et al. [107] continued with 45 deg skewed ribs. Mochizuki et al. [108] performed heat transfer measurements. Cheah et al. [109] performed LDA measurements on channels with strong curvature and found that the flow separates within the bend. Hwang and Kuo [110] derived heat transfer correlations. Iacovides et al. [111] performed
experiments on local heat transfer in rotating square ended U-bend. Liou and Chen [112] performed LDV study on smooth duct with 180 deg straight corner turn. Including in-line ribs in recent study, Liou et al. [95] performed flow and heat transfer measurements.

### 4.1.3. 2 Effect of Curvature-Predicting Curvature

Luo and Lakshminarayana [72] studied high Re non-rotating 2-D case using k-ε, NL k-ε, ASM and RSM turbulence models, with RSM provided the best predictions for major features including strong enhancement of turbulence near the concave wall, large reduction of turbulence near the convex wall, and separation down stream of the convex wall. The simulation with RSM showed that the mean flow inside the bend is nearly insensitive to the upstream inflow conditions. Variation of $\delta/R$ has more influence on the turbulence amplification near the concave wall than on turbulence damping near the convex wall. In [113] they concluded that for RSM to capture the large amplification of turbulence in concave boundary layers, the generation term in e-equation should not keep pace with that in the k-equation. Studying non-rotating channels, Rumsey et al., [114] concluded that the RSM turbulence model was able to capture the full extent of suppressed turbulence near the convex wall, while the one and two equation models and EASM were not. Zhang et al, [115] studied numerically the flow structure and friction factor for fully developed flow in parallel rotating rectangular ducts. They concluded that possibly four kinds of secondary vortices exists: due to centrifugal force, due to Coriolis force, Dean vortices due to the centrifugal instability and Coriolis vortices due to the Coriolis instability. For small De of square duct, no vortex due to the flow instability is obtained, while for high De Dean vortices appear for non-rotating cases. Besserman and Tanrikut [116] and Choi et al. [117] pointed out
the importance of resolving the near wall region in curved ducts. Iacovides et al. [118] used a
differential stress model (DSM), with and without modification to the $\varepsilon$-equation. The results
were improved slightly compared to results using ASM. Iacovides et al. [119] compared between
using different wall treatments with an ASM, either using one-equation model ($\varepsilon$ determined
from prescribed length scale) or a two-equation based model ($\varepsilon$ given from its transport
equation). The separated region inaccurately predicted, with only a slight improvement using the
full LRN- version of the model. In the study by Nikas and Iacovides [120] it was concluded that
for heat transfer predictions a LRN turbulent model is preferable to a zonal model in the near
wall region. The separation was however not well predicted using any of the turbulence models.
From numerical simulations in the Cheah et al. U-bend, Bredberg [90] used unmodified LRN
two-equation models. Length-scale corrections were needed, especially when predicting heat
transfer within the bend region. Using LRN k-omega model to simulate U-bend, Rigby et al.
[121] achieved reasonable accuracy even for heat transfer.

4.1.4 Effect of Rotation-Predictive Capabilities

In the 7th ERCOFTAC workshop a 2D rotating flow was used to evaluate turbulence models and
compared to DNS-data Kristoffersen and Andersson [122] for LRN turbulent flow. The results
showed that the isotropic EVMs are unable to capture the increase/decrease of turbulence kinetic
energy on the unstable/stable side. Better agreement was achieved when non-linear EVMs,
EARSM and RSMs were used, although discrepancies still persisted in the near wall region,
especially on the stabilizing side. Most turbulence models over-predicted the suppression of
turbulence, with a too early flow laminarisation. Notable improvements can be achieved if
rotational modified length-scale corrections-in analog to those used for stream curvature-are applied [90]. It has been shown that even the $k-\varepsilon-\nu^2-f$ model needs rotational modification to capture the skewed velocity profile in rotating ducts.

4.1.5 Numerical Simulations of Combined Effects Concerning Predictive Capabilities

The case with $Re = 25\,000$, $Ro = 0.24$ with staggered ribs skewed at 45 angle to the stream direction was investigated by several authors. In Bonhoff et al. [88] the number of nodes was reduced through employed wall-functions using $k-\varepsilon$ turbulence model and RSM. They used the commercial code Fluent, which was able to predict the Nusselt number trends correctly although the magnitudes of the changes were inaccurate. Stephens et al. [82] achieved better agreement using LRN $k-\omega$ turbulence model in a SST turbulence model with an increase in the number of nodes to $760 \times 65 \times 65 = 3.2 \times 10^6$. In Jang et al., [123] they used a single leg of the U-bend to reduce the number of nodes. Their grid refinement study with the used LRN version of RSM showed a significant difference (20%) in the predicted heat transfer between a mesh with $21 \times 21$ nodes and the one with $41 \times 41$ nodes in the cross-section plane. The $61 \times 61$ refinement gave a difference of 4%. Using the $570 \times 41 \times 41 = 9.6 \times 10^5$ mesh they achieved good agreement with the measured data. The result showed a significant change in the heat transfer levels due to the rotational induced Coriolis forces, with a reduction of up to 40% on the leading face, and an increase on the trailing face by up to 35%. Iacovides and Raisee [106] studied (experimentally and numerically and also by Bredberg [90]) the ribbed case equivalent to the smooth case studied by Cheah et al. Through adoption of zonal turbulence model ($k-\varepsilon$ and RSM) and the usage of bounded QUICK discretization scheme Iacovides and Raisee [106] could reduce the
computational requirements. The results showed the advantage of using RSM in regions of flow separation, however the main flow features still captured by the k-ε model. For negative rotation case, with the rotation and curvature induced force opposed each other, the agreements were significantly reduced as compared to the case with positive rotation. The cause may be the inability of zonal models to accurately assess these phenomena within the near wall region. Predberg, [90] made a numerical study of the same case, Re = 95 000, Ro = 0.2, with normal ribs arranged staggered, using a modified LRN $k-\omega$ turbulence model. It was found that the cross-diffusion terms were numerically problematic and an additional modification was necessary to limit these terms. Similar to previous studies discrepancies were found within the U-bend and immediately downstream the bend, however overall both the trends and magnitude were in good agreement with measurements.

Research relevant to predicting heat transfer in internal cooling channels is not as extensive as the experimental research. Dunn [124] gave a brief review of experimental and predicting capability work. Rigby [125] used k-ω turbulence model with concentration on grid structure to describe heat transfer in a ribbed channel with 180 deg bend and showed reasonable agreement with experimental results of Park et al. [126]. Rigby [121] performed three-dimensional simulation in a rectangular duct with 180 deg bend using multi-block and single-block grids with k-omega model. For the same number of cells, the multi-block grid predictions were more accurate when compared to experimental results by Arts et al. [127]. Using the same k- ω model, Rigby et al. [128] performed numerical simulation to predict the heat transfer in a straight square sectioned duct, but with three smooth walls and with ribs and bleed holes on the fourth wall. The prediction was in good agreement with the experimental results of Ekkad et al. [129]. Murata and
Mochizuki [130] studied the effect of centrifugal buoyancy on turbulent heat transfer using LES in rotating square duct with 90 and 60 deg ribs. Their results show that in the 60 deg rib case the friction factor decreased by increasing Gr, which was opposite to the results of the 90 deg case. The buoyancy-induced aiding flow enhanced and suppressed the Coriolis induced secondary flow in the radially outward and inward flow configurations, respectively. The predictions by Rathjen et al. [94] using k-e model with wall function and a 2-layer approach shows significant discrepancies compared to their heat/mass transfer experimental results. Jang et al. [123] used RSM to predict flow and heat transfer for the experimental case studied by [107] using one pass with Ro up to 0.24. They concluded that a second-moment closure is necessary to predict the high anisotropy produced by secondary flows that induced by the angle ribs, rotating buoyancy and Coriolis forces.

4.2 Work with Reference to Parallel Mode Rotation

Air and Liquid coolants are used in generator internal cooling. Researchers have investigated heat transfer parameters using various types of coolants flowing in channels with Re (Reynolds number) ranging from 2,000 to 37,000. Table 4.1 (see Morris [131]) depicts the investigations along with the corresponding range of parameters.
Table 4.1 Experimental Investigations on Heat transfer and Turbulence in Parallel Mode Rotating Ducts.

<table>
<thead>
<tr>
<th>Authors</th>
<th>Topic</th>
<th>Duct cross-section</th>
<th>RP M</th>
<th>Re</th>
<th>Final outcome</th>
<th>L/Dh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Morris D., 1964</td>
<td>HT Laminar heated flow</td>
<td>Circular</td>
<td>300</td>
<td>10,000</td>
<td>Nu, Cf</td>
<td>24</td>
</tr>
<tr>
<td>Davies and Morris D., 1965</td>
<td>HT Laminar heated flow</td>
<td>Circular</td>
<td>300</td>
<td>10,000</td>
<td>Nu, Cf</td>
<td>24</td>
</tr>
<tr>
<td>Humphreys, 1966</td>
<td>Entrance Turbulent</td>
<td>Circular</td>
<td>300</td>
<td>20,000</td>
<td>Nu, Cf</td>
<td>48</td>
</tr>
<tr>
<td>Humphreys and Morris, 1967</td>
<td>Entrance Turbulent</td>
<td>Circular</td>
<td>300</td>
<td>20,000</td>
<td>Nu, Cf</td>
<td>6</td>
</tr>
<tr>
<td>Le Feurre, 1968</td>
<td>Entrance Turbulent</td>
<td>Circular</td>
<td>300</td>
<td>20,000</td>
<td>Nu, Cf</td>
<td>2</td>
</tr>
<tr>
<td>Morris D., 1968, 1970</td>
<td>HT Laminar heated flow</td>
<td>Circular</td>
<td>300</td>
<td>10,000</td>
<td>Nu, Cf</td>
<td>24</td>
</tr>
<tr>
<td>Jakamota and Fukui, 1971</td>
<td>HT Laminar heated flow</td>
<td>Circular</td>
<td>2700</td>
<td>31,900</td>
<td>Nu, Cf</td>
<td>10</td>
</tr>
<tr>
<td>Woods and Morris, 1974</td>
<td>HT Laminar heated flow</td>
<td>Circular</td>
<td>1500</td>
<td>2,500</td>
<td>Nu, Cf</td>
<td>24</td>
</tr>
<tr>
<td>Woods, 1975</td>
<td>HT Laminar heated flow</td>
<td>Circular</td>
<td>1000</td>
<td>2,500</td>
<td>Nu, Cf</td>
<td>28</td>
</tr>
<tr>
<td>Skiadareissis and Spalding, 1976</td>
<td>Entrance Laminar</td>
<td>Circular</td>
<td>1000</td>
<td>2,500</td>
<td>Nu, Cf</td>
<td>24</td>
</tr>
<tr>
<td>Morris and Woods, 1978</td>
<td>Entrance Turbulent</td>
<td>Circular</td>
<td>1000</td>
<td>20,000</td>
<td>Nu, Cf</td>
<td>48</td>
</tr>
<tr>
<td>Nukayama and Fuzoika, 1978</td>
<td>Dev. Turbulent</td>
<td>Circular</td>
<td>1000</td>
<td>37,000</td>
<td>Nu, Cf</td>
<td>27.3</td>
</tr>
<tr>
<td>Morris, 1980, 1981</td>
<td>Dev. Laminar, isothermal, incompressible</td>
<td>Circular</td>
<td>600</td>
<td>10,000</td>
<td>Nu, Cf</td>
<td>24</td>
</tr>
<tr>
<td>Dias, 1978 &amp; Morris and Dias, 1981</td>
<td>Exp. Developed laminar HT, air</td>
<td>Square</td>
<td>300</td>
<td>2,000</td>
<td>Nu</td>
<td>51</td>
</tr>
<tr>
<td>Dias, 1978 &amp; Morris and Dias, 1981</td>
<td>Exp. Laminar HT in the entrance region, air</td>
<td>Square</td>
<td>300</td>
<td>2,000</td>
<td>Nu</td>
<td>32</td>
</tr>
<tr>
<td>Dias, 1978 &amp; Morris and Dias, 1981</td>
<td>Exp. Turbulent HT, air</td>
<td>Square</td>
<td>600</td>
<td>10,000</td>
<td>Nu</td>
<td>32</td>
</tr>
</tbody>
</table>
Literature review reveals that most of these studies have concentrated on laminar Nu (Nusselt number) and $C_f$ (friction factor) for a range of rotational speeds from 300 rpm to a maximum of 2700 rpm. Generator rotors, in practical applications can rotate at as much high speed as 90,000 rpm. The flow field and heat transfer at such high speeds have different characteristics; as will be shown in the results section. One of the objectives of this research is to explore the flow phenomena and heat transfer in square sectioned channels rotating at high rotation and buoyancy numbers.

Morris [131] provides theoretical, experimental and numerical data for circular cross section rotating channels for both laminar and turbulent flows. The available data in literature concerning rotating square channels is so limited. Morris [131] also provides heat transfer measurements in square channels for low rotation numbers in the range from 0.0125 to 0.063 and with maximum constant heat flux of 6.45 kw/m2 for wall boundary condition. Majumdar et al. [132] described theoretically some aspects of flow and heat transfer. Some agreement with measured Nusselt number was reported without providing precise details. The work by Dias [133] was extended by Morris and Dias [134] to include an experimental study of turbulent flow with air as a coolant. They found that for fixed Reynolds number, fixed eccentricity and for a given constant heat flux, the heat transfer improved with increases in the rotational speed. They used the rotational Reynolds number, $J_b$ instead of Ro to express rotational effects. Their results showed that at each rotational speed, the local Nusselt number distributions had similar trends. Near the channel entrance, the initially high region of heat transfer asymptotes towards a plateau region consistent with boundary layer development that occurs in a heated tube. Near the channel exit, the local heat transfer increases again due to the combination of the exit type effects and end
conduction type. It was noted that there is a systematic improvement in the local heat transfer with increasing rotational speed for the rotational speed range studied. There was also a slight tendency for the Nusselt number to increase with increases in heat flux. Which suggested that there was still a buoyancy effect even in turbulent flow regime. Morris and Dias [134] also derived the following correlating equation for the mean Nusselt number:

$$\text{Nu}_m = 0.012 \text{Re}^{0.78} \text{J}_b^{0.1}$$ (4.1)

where $\text{J}_b = \Omega b^2/2\nu$ and $b$ is the duct height

Equation 4.1 correlates data with a maximum scatter of $\pm 14\%$ and was derived from data covering the range $120 < \text{J}_b < 620$, $32 < \text{R/Dh} < 48$, $500 < \text{Re} < 20,000$ for $\text{L/Dh} = 48$. They compared the results using equation 4.1 with equation for circular channels derived by Morris and Woods [135] and with works of Humphreys (1966) and Le Feuvre (1968), see Morris [131]. It has been noted that the results of equation 4.1 are in better agreement than the results using circular channels correlation. Thus the data from circular cross section tubes is not recommended for square cross-sectioned channels.
CHAPTER FIVE: NUMERICAL APPROACH AND BOUNDARY CONDITIONS

In this dissertation FLUENT CFD-code is employed to perform the simulation. The numerical solution method used to solve N-S equations is the finite volume method. A control-volume-based technique is used to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis.

5.1 Discretization

Discretization of the governing equations can be illustrated by considering the steady-state conservation equation for transport of a scalar quantity \( \phi \). Consider the following equation written in integral form for an arbitrary control volume \( V \):

\[
\int \rho \phi \vec{v} \cdot d \vec{A} = \int \Gamma_\phi \nabla \phi \cdot d \vec{A} + \int S_\phi dV
\]  

(5.1)

where

\( \rho \) = density

\( \Gamma_\phi \) = diffusion coefficient of \( \phi \)
Equation 5.1 is applied to each control volume, or cell, in the computational domain. The two-dimensional, triangular cell shown in Figure 5.1 is an example of such a control volume. Discretization of Equation 5.1 on a given cell yields

$$
\sum_{f=1}^{N_{\text{faces}}} \rho_f v_f \cdot A_f = \sum_{f=1}^{N_{\text{faces}}} \Gamma_\phi (\nabla \phi)_n \cdot A_f + S_\phi V
$$

(5.2)

where $N_{\text{faces}}$ = number of faces enclosing cell

$\phi_f$ = value of $\phi$ convected through face

$$
\sum_{f=1}^{N_{\text{faces}}} \rho_f v_f \cdot A_f = \text{sum of mass fluxes through the face}
$$

$A_f$ = area of face

$(\nabla \phi)_n$ = magnitude of $\nabla \phi$ normal to face

$V$ = cell volume

The equations solved take the same general form as Equation 5.2 and apply readily to multi-dimensional, unstructured meshes composed of arbitrary polyhedra. The solver stores discrete values of the scalar $\phi$ at the cell centers ($C_0$ and $C_1$ in Figure 5.1). However, face values $\phi_f$ are required for the convection terms in Equation 5.2 and must be interpolated from the cell center values. This can be done using an upwind scheme. Upwinding means that the face value $\phi_f$ is derived from quantities in the cell upstream relative to the direction of the normal velocity $v_n$. The diffusion terms in Equation 5.2 are central-differenced and are always second-order accurate.
5.1. 1 Second-Order Upwind Scheme

Quantities at cell faces are computed using a multidimensional linear reconstruction approach [136]. In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. Thus the face value $\phi_f$ is computed using the following expression:

$$\phi_f = \phi + \nabla \phi \cdot \Delta \vec{s}$$  \hspace{1cm} (5.3)

where $\phi$ and $\nabla \phi$ are the cell-centered value and its gradient in the upstream cell, and $\Delta \vec{s}$ is the displacement vector from the upstream cell centroid to the face centroid. This formulation requires the determination of the gradient $\nabla \phi$ in each cell. This gradient is computed using the divergence theorem.
5.1.2 Discretization of the Momentum Equation

The discretization scheme described above for a scalar transport equation is also used to
discretize the momentum equations. For example, the X-momentum equation can be obtained by
setting $\phi = u$:

$$ a p u = \sum_{n_b} a_{n_b} u_{n_b} + \sum p_{f} A \cdot \vec{i} + S \quad (5.4) $$

If the pressure field and face mass fluxes were known, Equation 5.4 could be solved and a
velocity field obtained. However, the pressure field and face mass fluxes are not known a priori
and must be obtained as a part of the solution.

The solver uses a co-located scheme, whereby pressure and velocity are both stored at cell
centers. However, Equation 5.4 requires the value of the pressure at the face between cells $C_0$
and $C_1$. Therefore, an interpolation scheme is required to compute the face values of pressure
from the cell values.

5.1.3 Pressure Interpolation Schemes

Interpolating the pressure values at the faces can be done using momentum equation
coefficients [137]. This procedure works well as long as the pressure variation between cell
centers is smooth. When there are jumps or large gradients in the momentum source terms
between control volumes, the pressure profile has a high gradient at the cell face, and cannot be
interpolated using this scheme. This scheme will have trouble in flows with large body forces,
such as in strongly swirling flows and in high-Rayleigh-number convection considered in this
dissertation. In such cases, it is necessary to pack the mesh in regions of high gradient to resolve the pressure variation adequately.

Another source of error is that FLUENT assumes that the normal pressure gradient at the wall is zero. This is valid for boundary layers, but not in the presence of body forces or curvature. The alternate method is the PRESTO! (PREssure STaggering Option) scheme that uses the discrete continuity balance for a "staggered" control volume about the face to compute the "staggered" (i.e., face) pressure. This procedure is similar in spirit to the staggered-grid schemes used with structured meshes [138]. For flows with high swirl numbers, high-Rayleigh-number natural convection, high-speed rotating flows, flows involving porous media, and flows in strongly curved domains, the PRESTO scheme proved to obtain comparable accuracy and to enhance convergence. Thus in this dissertation PRESTO scheme is used for all calculations.

5.1.4 Discretization of the Continuity Equation

Equation 5.1 may be integrated over the control volume in Figure 5.1 to yield the following discrete equation

\[ \sum_{f} J_{f} A_{f} = 0 \]  

(5.5)

where \( J_{f} \) is the mass flux through face \( f \).

The momentum and continuity equations are solved sequentially, where the continuity equation is used as an equation for pressure. However, pressure does not appear explicitly in Equation 5.5 for incompressible flows, since density is not directly related to pressure. The SIMPLE (Semi-
Implicit Method for Pressure-Linked Equations) algorithms [138] is used for introducing pressure into the continuity equation. In order to proceed further, it is necessary to relate the face values of velocity, \( \vec{v}_n \), to the stored values of velocity at the cell centers. Linear interpolation of cell-centered velocities to the face results in unphysical checker-boarding of pressure. FLUENT uses a procedure similar to that outlined by Rhie and Chow [137] to prevent checker-boarding.

5.1.5 Density Interpolation Schemes

For incompressible flows, the solver uses arithmetic averaging of density at cell faces.

5.1.6 Pressure-Velocity Coupling

The SIMPLE algorithm is used for pressure-velocity coupling. SIMPLE uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

For relatively uncomplicated problems (laminar flows with no additional models activated) in which convergence is limited by the pressure-velocity coupling, a converged solution may be obtained more quickly using SIMPLEC. With SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which aids in convergence speed-up. In some problems, however, increasing the pressure-correction under-relaxation to 1.0 can lead to instability.
5.1. 7 Cell-Based Derivative Evaluation

In this approach, the face value, $\bar{\phi}_f$, is taken from the arithmetic average of the values at the neighboring cell centers, i.e.,

$$\bar{\phi}_f = \frac{\phi_{C_i} + \phi_{C_{i+1}}}{2}$$  \hspace{1cm} (5.6)

5.1. 8 Multigrid Method

A multigrid scheme is used to accelerate the convergence of the solver by computing corrections on a series of coarse grid levels. The use of this multigrid scheme can greatly reduce the number of iterations and the CPU time required to obtain a converged solution. Multigrid techniques allow global error to be addressed by using a sequence of successively coarser meshes. This method is based upon the principle that global (low-frequency) error existing on a fine mesh can be represented on a coarse mesh where it again becomes accessible as local (high-frequency) error: because there are fewer coarse cells overall, the global corrections can be communicated more quickly between adjacent cells. Since computations can be performed at exponentially decaying expense in both CPU time and memory storage on coarser meshes, there is the potential for very efficient elimination of global error.
5.2 Grid Generation and Boundary Conditions for Channels in Orthogonal Mode Rotation

In this dissertation experimental data from four sources are considered for comparison purposes and for validation. Thus the simulated geometry of the coolant channels, the boundary conditions and the generated grid are described next.

5.2.1 Rib-Roughened Channel – Stationary Case

The performance of turbulence models is evaluated in the in line rib-roughened stationary case of Rau et al. [49]. The numerical grid of the stationary case is shown in Figure 5.2 generated using GAMBIT grid generator. For enhanced wall treatment, the first cell next to a wall must be of order unity. To resolve the near wall viscous region 11-grid points were placed in the boundary layer near all walls. The shape of each rib needs at least 12 grid points to be resolved. The convergence criterion for all quantities error was 10E-4 and 10E-6 for the energy equation. At the inlet, the velocity and turbulence intensity profiles used were those measured by Rau. To reproduce periodicity of the flow a channel with five ribs was simulated and the data provided here was taken in between 4th and 5th ribs. The blockage ratio e/Dh was 0.1, and the pitch to rib height ratio is P/e=9. Reynolds number was fixed to 30,000 and uniform heat flux boundary condition is applied to all walls. The channel was made of 0.05x0.05m Plexiglas to allow visualization. The density of the fluid is approximated by \( \rho = \rho_o T / T \) to account for density variations caused by the temperature differences, while piecewise linear functions were used to account for the viscosity, thermal conductivity and specific heat properties variations.

The measured Nusselt numbers were normalized with Dittus-Boelter equation, [139]:

\[
\frac{Nu}{\sqrt{Re Pr}} = 0.023 Pr^{0.4} Re^{0.8} 
\]
\[ \text{Nu} = 0.023 \text{Re}^{0.8} \text{Pr}^{0.4} \] (5.7)

The experimental uncertainty in \( \text{Nu}/\text{Nu}_0 \) was estimated to be 5%.

A grid-refinement study was performed using RSM model for three different grid distributions of 40x40x500, 50x50x500 and 50x50x600 grid lines. The grid refinement in the streamwise direction showed negligible enhancement to the calculated Nusselt number compared to the measured of about 3%. In the cross-stream direction, a comparison between calculated and measured values of Nusselt number along a center line in the floor showed 4% maximum change in Nusselt number between 40x40x500 and 50x50x500 grid points. Thus all results presented here are based on 50x50x500 grid, which resulted in 1,250,000 grid points.

Figure 5.2 Numerical Grid for Stationary Channel with in-Line Ribs Studied Experimentally by Rau et al. [49]
5.2. 2 Smooth-Walls Channel – Rotating Case

The channel with smooth walls studied by Wagner et al. [75] was used in this study for comparison using two of the four legs with one U-turn. Enhanced wall treatment is used for the near wall treatment of the flow and heat transfer.

The geometry of the two-pass channel is shown in Figure 5.3. The four walls of the square duct are denoted as the leading, trailing, inner and outer sides. All walls are heated to a constant temperature. The coolant temperature is \( T_0 \) (i.e., \( \theta = (T - T_0) / (T_w - T_0) = 0 \)) at the duct entrance and the wall temperature was kept constant at \( T = T_w(\theta = 1) \) for all sidewalls. A uniform velocity profile was used at the entrance of the channel and a zero normal pressure gradient is set at the exit of the channel. The density of the fluid is approximated by \( \rho = \rho(T, T_w) \) to account for density variations, while piecewise linear functions were used to account for the viscosity, thermal conductivity and specific heat properties variations. In this study, the range of rotation numbers considered was 0.0, 0.215, 0.43, 0.86 and 1.29. Density ratios of 0.13, 0.229 and 0.4, which correspond to wall temperature of 344, 389 and 500 K respectively with Reynolds number fixed to 25000. Nusselt numbers were calculated based on the average bulk temperature and normalized with a smooth tube correlation (Kays and Crawford [140]) for fully developed, nonrotating, turbulent flow:

\[
Nu_y = 0.0176 Re^{0.8} \tag{5.8}
\]

In the following discussion for all figures, note that \( Y/D_h=0 \) corresponds to trailing surface, \( Y/D_h=1 \) corresponds to leading surface, \( X/D_h=0 \) corresponds to outer surface at location Q6, \( X/D_h=1 \) corresponds to inner surface at location Q6, \( X/D_h=3.5 \) corresponds to inner surface at
location N4, X/Dh=4.5 corresponds to outer surface at location N4, Z/Dh=0 corresponds to inner surface at location P5 and Z/Dh=1 corresponds to outer surface at location P5.

Figure 5.3 Geometry for Two Pass Square Channel Tested Experimentally by Wagner et al. [74]

Figure 5.4 shows the numerical grid generated using GAMBIT grid generator for this study. $y^+$ for the first cell next to a wall is taken to be of order unity. To resolve the near wall viscous region ten grid points were placed in the boundary layer near all walls. The minimum convergence criterion for all quantities error was 10E-4 and 10E-7 for the energy equation. A grid-refinement study was performed using three different grid distributions of 40x40x320, 40x40x420 and 52x52x320 grid points. Comparison between the calculations and measurements
was performed for rotation number of 0.24 and for coolant-to-wall density ratio \( \frac{\Delta \rho}{\rho} \) of 0.13, see Figure 5.5. The grid refinement in the axial direction of the bend region showed minor enhancement to the solution. A comparison between calculated and measured values of Nusselt number on the leading and trailing edge surface (for \( Re=25,000 \), \( R_o = 0.24 \), \( \Delta \rho/\rho = 0.13 \)) showed a maximum enhancement of 4% in Nusselt number between 40x40x320 and 40x40x420 grid distribution. Increasing the number of grid points on the cross-stream direction almost did nothing to the solution. Thus all results presented here are based on 40x40x320 grid points. The predicted and measured Nusselt number ratios are in good agreement except at the channel inlet. This is due to the difference between the experimental and assumed boundary conditions.

![Numerical Grid for Channel with Smooth Wall Studied Experimentally by Wagner et al.](image)

Figure 5.4 Numerical Grid for Channel with Smooth Wall Studied Experimentally by Wagner et al. [75]
5.2.3 Rib-Roughened Channel – Rotating Case

The results for four-leg square channel with staggered rib-roughened walls tested by Wagner et al. [85] were used in this study for comparison using two of the four legs with one U-turn. Enhanced wall treatment is used for the near wall treatment of the flow and heat transfer.

The geometry of the two-pass channel is shown in Figure 5.6. The four walls of the square duct are denoted as the leading, trailing, inner and outer sides. Eleven rounded ribs with fillet were placed on leading and trailing surfaces of every passage in a staggered arrangement with ribs on the leading surface offset upstream from those on the trailing surface by a half pitch (P). The length of every passage is 14*Dh, the U-turn inner diameter (2*r) is 2.5*Dh, and the distance from the axis of rotation to channel inlet (R) is 44*Dh duct. The distance from the duct inlet to the first rib and from the duct outlet to last rib is 3.5*Dh. The rib height-to-hydraulic diameter (blockage ratio) is 0.1 and the rib-pitch-to-height ratio is 10. In this study, rotation numbers considered are 0.0, 0.238, 0.475, 0.74 and 1. Density ratios of 0.13, 0.229, and 0.3 which
correspond to wall temperature of 344, 389, and 428K, respectively with Reynolds number fixed to 25000 and operating pressure to 10-bar. All walls are heated to a constant temperature. The coolant temperature is $T_0$ (i.e., $\theta = (T - T_0)/T_w - T_0 = 0$) at the duct entrance and the wall temperature was kept constant at $T = T_w (\theta = 1)$ for all sidewalls. Nusselt numbers were normalized with a smooth tube correlation (Kays and Crawford [140]) for fully developed, nonrotating, turbulent flow, Equation 5.8.

Figure 5. 7 shows the rib geometry and the numerical grid generated using GAMBIT grid generator for this study. Uniform velocity profile was used at the entrance of the channel. The density of the fluid is approximated by $\rho = \rho \cdot T_T / T$ to account for density variations caused by the temperature differences, while piecewise linear functions were used to account for the viscosity, thermal conductivity and specific heat properties variations. $y^+$ the first cell next to a wall is of order unity. To resolve the near wall viscous region ten grid points were placed in the boundary layer near all walls. The shape of each rib needs 9 grid points to be resolved. The convergence criterion for all quantities error was 10E-4 and 10E-7 for the energy equation. A comparison between the calculated and measured Nusselt number ratio was performed for rotation number of 0.0, 0.238 and 0.35 at coolant-to-wall density ratio ($\Delta\rho / \rho$) of 0.13 and is given in Figure 5. 8. A grid-refinement study was performed using three different grid distributions of 44x44x880, 44x44x 960 and 52x52x880 points. The grid refinement in the streamwise direction of the passages and the bend region showed minor enhancement to the solution. A comparison between calculated and measured values of Nusselt number on the leading surface (for $Re=25,000$, $Ro = 0.238$, $\Delta\rho / \rho = 0.13$) showed 3% maximum change in
Nusselt number between 44x44x880, 44x44x960 grid distribution. Increasing the number of grid points on the cross-stream direction from 44 to 52 did not enhance the solution further. Thus all results presented here are based on 44x44x880 grid distribution, which resulted in 1,703,680 grid points.

Figure 5. 6 Geometry for Two Pass Square Channel with Staggered Ribs Studied Experimentally by Wagner et al. [85]
The predicted Nusselt number using RSM with enhanced near wall treatment are in good agreement with measured data, Figure 5. 8. At the entrance, the predictions for all cases are less than the measured. This may be because of the difference between the experimental and numerical inlet boundary conditions.

Figure 5. 8 Predicted and Measured, [85] Nusselt Number Ratios on Leading Surface; Re=25,000, DR=0.13

80
Similar geometry to that studied by Morris [131] is chosen for the simulation. The channel is of square cross section with length to hydraulic diameter ratio, $L/D_h = 64$ and the eccentricity parameter, $R/D_h = 32$. The inlet Reynolds number is fixed to 20,000 and uniform velocity profile is assumed at the inlet, while zero normal pressure gradient is set at the channel exit. Rotation numbers considered are 0.0613, 0.14 and 0.3, which correspond to rotational Reynolds number, $J_b$ of 628, 1386 and 3472. Uniform wall heat flux boundary condition is applied for three cases of $q_1'' = 6.45 \text{ w/m}^2$, $q_2'' = 9,000 \text{ w/m}^2$ and $q_3'' = 12,000 \text{ w/m}^2$. The highest rotational Reynolds number, $J_b$ and wall heat flux studied by Morris 1 were 628 and 6.45 w/m², respectively. The experimental data given in Morris [131] was not enough to compare results from this study to experimental results. However a comparison is provided in chapter six with stationary ($Ro=0$) and rotating ($Ro=0.063$) cases.

Figure 5. 9 gives the numerical grid generated using Gambit for this simulation. A grid of $44 \times 44 \times 700 = 1,355,200$ grid points is used for this simulation with $y^+ \ll 1$ the first cell next to a wall is less than unity. Eleven grid points were placed near all walls to capture the flow inside the boundary layer.
Figure 5. 9 Numerical Grid for Square Channel in Parallel Mode Rotation
CHAPTER SIX: RESULTS AND DISCUSSION

6.1 Comparison Between Two-Equation EVMs and RSM Turbulence Models

The predicting performance of turbulence models, outlined in chapter two, is evaluated using experimental data from the stationary rib-roughened case of Rau et al. [49] and the rotating rib roughened case of Wagner et al. [85].

6.1.1 Stationary Rib-Roughened Channel

Figure 6.1 compares the heat transfer enhancement along vertical line at a distance from the last rib on the smooth side wall predicted using five two-equation and RSM turbulence models. The predictions using RSM are the most accurate within 5%. The standard k-ω and the SST k-ω predicted the trend correctly but overpredicted Nu/Nuo near the rib top by 16% and 25%, respectively. The worst were predictions by all 3 k-ε models. The trend was predicted incorrectly by standard k-ε model near the rib top, but a slight enhancement was achieved using the RNG model. The Realizable model could not predict the correct trend in the vertical direction between the ribs.
In Figure 6.2 Nusselt number predictions along the centerline of the floor in between the ribs show that k-ε models performed better than the two k-ω models, while RSM predictions are more accurate.

Figure 6.1 Heat Transfer Enhancement along a Vertical Line on the Smooth Wall-Stationary Case at Distance e from the Last Rib

Figure 6.2 Heat Transfer along the Centerline of the Floor in Between the Ribs-Stationary Case
All models predicted the correct trend of the streamwise velocity as shown in Figure 6. 3. Again RSM predictions are the most accurate, while the two k-\omega models are the worst especially in the regions of separation and maximum velocities. All three k-\epsilon models overpredicted the separation region behind the rib and underpredicted the maximum velocity in front of the rib.

Figure 6. 3 Streamwise Velocity at Y/e = 0.1 in Symmetry Plane

Figure 6. 4 Flow Entrainment in Between the Ribs at Y/e = 1 in Symmetry Plane
The vertical velocity component shown in Figure 6.4 was predicted correctly by all models with slight underprediction in the separation region and overprediction shortly before the last rib. However, k-ω models performed worst.

6.1.2 Rotating Rib-Roughened Channel

Predicted Nusselt number using two-equation models and RSM are compared with experimental results from Wagner et al. for rib-roughened rotating channel. As shown in Figure 6.5, RSM predictions of measured Nusselt number are accurate to within 9%. Thus in order to provide a quantitative comparison, RSM predictions of local Nusselt number are compared to two-equations models predictions assuming that RSM predictions are the most accurate.

![Graph showing Nusselt number ratios](image)

Figure 6.5 Predicted and Measured, [85] Nusselt Number Ratios on Leading Surface; DR=0.13.

Local Nusselt number was calculated using all six models at the centerline of leading and trailing surfaces at different axial locations S/Dh. The results are shown in Figure 6.6 and Figure 6.7, respectively.
In the first pass of leading surface all three k-ε models predicted higher Nu than RSM while the two k-ω models give lower Nu than RSM but the standard k-ω model changed this trend at S/Dh=12.4 to give the maximum Nu. In the U-tern RSM Nu was the maximum followed by SST k-ω, RNG k-ε then the Realizable k-ε models. The standard k-ε and k-ω models underestimated
RSM Nu by 33%. In the second pass all models predicted higher Nu than RSM at S/Dh=21.1 especially the standard k-ω and k-ε models and lower Nu than RSM at S/Dh=28.8 especially the standard k-ε model.

In the trailing surface, the predictions of local Nu were very close using all models except an underestimation by k-ω model at S/Dh=4.7 and S/Dh=21.1 and an overestimation at S/Dh=8.5 by RNG k-ε and SST k-ω models.

Figure 6.8 Streamwise Velocity Profile. Ro=0.238, DR=0.13, Re=25,000

The streamwise velocity profile in between the last two ribs (Z/P =0.75) using the 6 models is shown in Figure 6.8. There are no experimental data for the velocity profile to compare with. However the velocity profile predictions are compared to those studied by Bredberg who used different from this study experimental data from Icovides et al. to do the comparison with square cross section ribs, Ro=0.2, Re=100,000 and with uniform heat flux boundary conditions instead of constant temperature used in this study.
The skewness of the velocity profile towards the trailing surface because of the Coriolis force is well predicted by all models except the standard k-ω model that predicted early and excessive separation and shifted the velocity profile toward the leading surface. The maximum W/Wb of 1.35 was predicted by RSM at Y/Dh=0.25, which is very close (considering the differences between the cases studied) to Bredberg’s that was 1.44 at Y/Dh=0.27, which suggests that RSM velocity prediction is the most accurate.

Turbulence Anisotropy. The complexity of the flow inside internal cooling channels (ribs, U-turn, rotation, buoyancy etc.) generates turbulence anisotropy. Thus the turbulence model employed to solve such flow phenomena must be capable of capturing anisotropic turbulence effects. All two-equation EVMs are based on isotropic flow assumption; hence the only model that capable of predicting anisotropy is RSM. Figure 6. 9 depicts the ratio of the Reynolds stress normal components in the first pass, U-turn, and second passes using RSM. In the first pass, the Coriolis force is directed toward the trailing surface such that the streamwise velocity profiles are skewed toward the trailing wall. The rib generated shear layer on the trailing side and the high velocity gradients associated with skewed velocity profiles caused high turbulence anisotropy, approximately having values of 0.9-3 at Ro=0.24 and DR=0.13. In the second pass, the turbulence anisotropy has values of 0.6-2.2. In the U-turn, the turbulence anisotropy has values of 0.6-3. As a comparison, Liou et al. [95] studied the flow and heat transfer in a rotating two-pass square channel with 90 deg in-line square ribs on leading and trailing surfaces for Ro=0.2. They reported turbulence anisotropy, also in terms of the ratio of the Reynolds stress normal components, of 1.8-2.2, 1.8-2.8 and 1.3-2.4 in the first pass, U-turn, and second pass,
respectively. Considering the differences in boundary conditions and rib cross-section and arrangement, the anisotropy prediction using RSM in this study is reasonable.

Figure 6. 9 Turbulence Anisotropy in Terms of \( w'w'/v'v' \) Contours in the Centerline of the First and Second Passes and in Terms of \( u'u'/v'v' \) in the U-turn. Ro=0.238, DR=0.13, Re=25,000. Predicted Using RSM
Secondary flow. The predicted secondary flow vectors using all 6 models are shown in Figure 6. There are no available experimental data to compare the performance of the turbulence models in predicting secondary flow. However the flow configuration in the U-turn for rib-roughened channel does not change significantly from that for channels with smooth walls. Thus it is possible to compare the secondary flow in the U-turn from this study to Liu et al. study for the same Ro, Re, Dh, and DR.

In the first pass at S/Dh=12.4 RSM predicted two large Coriolis induced vortices near trailing surface and two small turbulence induced vortices at the corners near the leading surface. The three k-ε models also predicted the Coriolis induced vortices but their location was shifted from the trailing surface, however the turbulence-induced vortices cannot be seen clearly. The standard k-ω model predicted four vortices instead of two in the center of the channel, while the SST k-ω model predicted two vortices also in the center of channel near the outer and inner surfaces.

In the U-turn, where the effect of high-pressure gradients and cross-stream induced Coriolis dominates, RSM predicted large vortex near the trailing-inner surfaces and a smaller vortex near the trailing-outer surfaces in addition to the corner vortices. These predicted vortices do agree well with the predictions from Liu et al. The two k-ω models also predicted these vortices but with different size and location. The RNG k-ε model could predict the large two vortices but failed to predict the corner vortices. The worst predictions were those using the standard k-ε model followed by the Realizable k-ε model where the smaller vortex near the trailing outer surface barely can be seen. In the second pass at S/Dh=21.1 RSM predicted three large vortices near outer-leading, outer and inner-trailing surfaces respectively. At this location the effect of the
U-turn, Coriolis and buoyancy forces are interacting to cause significant flow disturbance. There is no experimental data available for comparison, but it is worth to state that all two-equation models predicted completely different secondary flow structure. The two k-ω models predicted two large vortices in addition to the corner vortices and similarly did the Realizable and RNG k-ε models. The standard k-ε model predicted only one large vortex in the center of the channel.
Figure 6. 10 Secondary Flow vectors (colored by streamwise velocity) in First Pass (S/Dh = 12.4), U-turn and Second Pass. Ro=0.238, DR=0.13, Re=25,000
6.1.3 Discussion on the Performance of Turbulence Models and the Assumptions Involved

6.1.3.1 Assumptions Involved in Closing Two-Equation Models

The results showed the advantages of RSM against the two-equation models. The reason is mainly the assumptions involved in closing the two-equation models. While the two-equation models are complete in that no new information is needed, they are to some degree limited to flows that do not depart so much from their fundamental assumptions.

Two-equation models, assume the local equilibrium, where turbulent production and dissipation given by k-ε equation are equal locally, which implies that the scales of turbulence are locally proportional to the scales of the mean flow. Therefore the two-equation models predicted incorrect trends when applied to non-equilibrium flow studied in this dissertation. Local equilibrium assumption follows from the fact that the Reynolds stresses must be estimated at every point in the flow field. Hence, the eddy viscosity is defined to be the proportional constant between the Reynolds stresses and the mean strain rate. This is the essence of the Boussinesq hypothesis. Since the turbulence and mean scales are proportional, the eddy viscosity can be estimated based on dimensional reasoning by using either the turbulent or mean scales. Thus, for k-ε model \( \mu \propto \rho k^2 / \varepsilon \) and for k-ω model \( \mu \propto \rho k / \omega \). When production does not balance dissipation, as the case studied here, then the Reynolds stresses to the mean strain rate is not a local constant and \( \mu \) should be a function of both turbulent and mean scales. Based on local equilibrium assumption, while the transport effects are included for turbulent scales they are
neglected for turbulent Reynolds stresses. In fact Reynolds stresses depend on the local conditions with some history effects. RSM accounts for these history effects, while two-equation models do not. The flow inside internal cooling channels cannot be assumed in local equilibrium because the flow is three dimensional, with sudden changes in mean strain rate, with U-turn, secondary flows, with separation and with rotating and stratified coolant, and hence the two-equation models failed.

The second assumption made by two-equation models is that the turbulent fluctuations, $U'$, $V'$, $W'$ are locally isotropic or equal. This is true of the smallest eddies at high Re, but the large eddies are in a state of steady anisotropy due to the strain rate of the mean flow. This assumption results in equal normal stresses at a point in the flow field. For the case studied in this dissertation, RSM predictions showed that this is not true, hence the two-equation models again failed.

The $k$-$\varepsilon$ equation is based on $\varepsilon$ equation, which actually represents the mechanism of the smallest eddies that physically accomplishes the dissipation. What is actually needed in the model is a length or time scale relevant to the large, energy containing eddies that are responsible for most of the turbulent stresses and fluxes. This leads to questions about how relevant the exact dissipation equation is, when the desired quantity is a length scale, characteristic of the large eddies.

The $k$-$\omega$ model solves for only the rate at which the dissipation occurs; $\omega \propto \varepsilon / k$. The equation governing $\omega$ has traditionally been formulated based on physical reasoning in light of the processes normally governing the transport of a scalar in a fluid.
6.1.3. 2 Closure Coefficients Determination

Usually the coefficients are determined by setting values such that the model obtains reasonable agreement with experiments. This approach is also questionable, as the constants determined for one application is not necessarily the best for the case studied in this dissertation, where no experimental data are available for high rotation and density ratio flow to set the constants against. Moreover, even if the constants are good for low level of rotation and density ratios, it is not necessarily that they are the best for high level of rotation and density ratios.

6.1.3. 3 Flows with Adverse Pressure Gradient

For flows with adverse pressure gradient, the backward facing step and flow inside rib-roughened channels, k-ω model proved to give superior performance to k-ε model. Wilcox attributes this to the large turbulence length scale predicted by the k-ε model in near-wall region. This can be improved by assigning the correct length scale as prescribed by Van Driest.

6.1.3. 4 Advantages of RSM with Enhanced Near Wall Treatment

RSM with enhanced wall treatment that resolves the sublayer region attempts to account for history effects, as the Reynolds stress equation includes convection and diffusion terms for the stresses and eliminate the need for equilibrium assumption and local isotropy. Also, the Reynolds stress equations include production and body force terms that can respond to the
effects of the streamline curvature, rotation and buoyancy. The two-equation models need corrections for streamline curvature, buoyancy and rotation.

Thus, as RSM predictions are superior to two-equation models predictions, all results presented in the following sections are predictions using RSM.

6.2 Effect of Coriolis and Centrifugal Forces on Turbulence and Transport at High Rotation Numbers and Density Ratios in Internal Cooling Channels with Smooth Walls – Results Using Standard Wall Functions

The effect of increasing Ro and DR on heat transfer and turbulence is studied by the author in [141] using standard wall functions approach for near wall treatment. Although the predictions are not accurate enough, but the Nu trend is still captured reasonably well.

6.2.1 Effect of Combined High Rotation and Density Ratio on Nusselt Number

Figure 6.11, Figure 6.12 and Figure 6.13 show the effect of increasing Ro on Nn/Nuo for DR of 0.13, 0.229 and 0.4. Nusselt number increases up to 5.4 times at S/Dh equals 10.5 in the leading side of the first pass, Figure 6.11. Increasing DR enhances the heat transfer rate at low Ro, but at high Ro does not. On the trailing surface, Nusselt number ratio increased up to 7.7, while for high density ratio the increase is slightly less. On the leading side of the U-turn at S/D of 17.5, Figure 6.12, the increase of Nusselt number ratio of up to 4.6 on leading side and 2.8 on trailing side is less than the increase in the first pass. The heat transfer rate on the leading surface decreases as DR increases, while on the trailing surface it enhances. Finally in the second leg at S/D of 24.5, Figure 6.13, the increase of Nusselt number ratio of up to 2.4 on leading side and
1.6 on trailing side is much less than on the first leg and the U-turn. These results show that at high (compared to low) Ro and density ratio, Nusselt number decreased slightly in both leading and trailing sides of the first pass, decreased slightly on leading side and increased on the trailing side of the U-turn, and increased on the leading side and decreased in the trailing side of the second pass. It is clearly shown that at high rotation numbers, increasing density ratio is not increasing Nusselt number any more, while the results in [74] showed that Nusselt number is always increasing as density ratio increases for the ranges of rotation numbers and density ratios studied.

![Figure 6. 11 Nusselt Number Ratio in the First Pass](image-url)
Figure 6. 12 Nusselt Number in the U-turn

Figure 6. 13 Nusselt Number in the Second Pass
6.2. 2 Comparison Between Results Using Standard Wall Functions and Enhanced Near Wall Treatment Approaches.

The treatment of the near wall region is very important in such complex flow phenomena considered in this dissertation. Thus, results for turbulent quantities using both standard wall functions and enhanced near wall treatment approaches, outlined in chapter three, are compared.

Using standard wall functions as opposed to enhanced near wall treatment, the normal stresses were predicted incorrectly in the boundary layer near the wall, Figure 6.14. This incorrect prediction affects the maximum normal stresses in the core region, where higher values than when enhanced near wall treatment is used were predicted. The shear stresses using the two approaches are shown in Figure 6.15. Again the predictions inside the boundary layer using standard wall functions are incorrect in the close to walls regions, especially near the leading surface. As an illustration, at Y/Dh=0.94 (near leading) the prediction of V’W’/Wo using standard wall functions is 0.25 compared to 0.1 predicted using enhanced near wall treatment approach. Thus, all predictions in the following subsections are performed using enhanced near wall treatment approach.
Figure 6. 14 Reynolds Normal Stress, $W'W'$ Using Standard Wall Functions (top) and Enhanced Near Wall Treatment (bottom)
Figure 6. 15 Reynolds Shear Stress, $V'W'$ Using Standard Wall Functions (top) and Enhanced Near Wall Treatment (bottom)
6.3 Effect of Coriolis and Centrifugal Forces on Turbulence and Transport at High Rotation Numbers and Density Ratios in Internal Cooling Channels with Smooth Walls – Results Using Enhanced Near Wall Treatment Approach

The experimental case of Wagner et al. [74] is considered to study the effect of increasing Ro and DR on turbulence and heat transfer. The results from this study are also published, see [142].

6.3.1 Effect of High-Rotation Number on Nusselt Number

In order to study the effect of increasing rotation number and density ratio on heat transfer rate, Nusselt number is calculated at 9 locations on the leading and trailing surfaces, for Ro of up to 1.29 and DR of up to 0.4. Note that this approach gives a comparison between different levels of Ro and DR. Calculation of Nusselt number is based on the mass weighted average bulk temperature at every section of the 9 locations. The results are shown in Figure 6.16. On the leading and trailing surfaces of the first and second passes, Nu always increases as Ro increases and slightly increases on the first pass leading and trailing surfaces as DR increases to 0.25 then decreases as DR increases to 0.4. In Wagner et al. [74] the results for rotation numbers of up to 0.475 showed that Nusselt number in trailing surface of the first pass increased by more than a factor of 3.5 and by a factor of 1.5 in the leading surface of second pass. These results are consistent with results shown in Figure 6.16. For higher rotation numbers studied here, on the leading surface of the first pass Nusselt number ratio increases up to 1.9 and 2.5 for Ro of 0.86 and 1.29 respectively then decreases before the U-turn and increases in the U-turn and again decreases for S/D up to 24.5 then increases slightly near the exit.
On the trailing side of the first pass high rotation number has increasing effect on Nusselt number ratio to more than 4.5 times for S/D of up to 7. The increase in Nu/Nuo has general trend as Ro changes. The trend is changed as DR increases to 0.25, where the heat transfer rate first enhances on the first pass up to S/Dh=14 then decreases or slightly increases on the U-turn and second pass. Increasing DR to 0.4 caused a decrease in heat transfer rate in all locations for the same Ro of 1.29. It is clearly shown that under the conditions of high Ro and DR the heat transfer trends differ from the case with low rotation numbers. In order to understand this
behavior the flow field and heat transfer at a vertical middle line of three planes: in the first pass, in the middle of the U-turn and in the second pass, are studied further. The effect of high-density ratios is addressed as well.

6.3. 2 Velocity and Temperature Profiles

Figure 6. 17 and Figure 6. 18 show the streamwise velocity and Figure 6. 19 gives the temperature distribution. At the entrance of the U-turn, High Ro increases the rapid acceleration of the colder fluid near the trailing surface (increase in Nu) and the separation of the hotter fluid near the leading surface (increase in Nu). Then increasing the DR enhanced this behavior but tended to decrease Nu because the thermal boundary layer near the walls increases and hence the temperature difference (Tw-Tb) increases. In the horizontal direction between inner and outer surfaces the gradients in temperatures and velocities are so small. The mixing of the fluid is better than for low rotation numbers. At the center of the U-turn the flow separates at Ro =1.29 and DR up to 0.4. In the horizontal direction the flow is heated uniformly having higher velocity magnitudes near the outer surface (slightly higher Nu) and separates at Dh/3 from the inner surface (slightly lower Nu) for high Ro and DR, which causes a high-pressure gradients. At the exit of the U-turn, High Ro and DR increase the velocity of the slightly hotter fluid near the leading surface (increase in Nu), and also slightly increase the velocity (increase in Nu) near the trailing surface, and then increasing the DR enhanced this behavior but tended to decrease Nu because the temperature difference (Tw-Tb) increases. In the horizontal direction between inner and outer surfaces the variations in temperatures and velocities are so small. The mixing of the fluid is better than for low rotation numbers.
Figure 6.17 Streamwise Velocity Profile in Vertical Direction
Figure 6.18 Streamwise Velocity Profile in Horizontal Direction
Figure 6. 19 Temperature Distribution
6.3. 3 Secondary Flow

Figure 6. 20 shows the secondary flow for different rotation numbers and density ratios at the center (location P5) and exit (location Q6) of the U-turn. At the center of U-turn Prandtl’s secondary flow of the first kind (generated by inviscid effects) due to rotational Coriolis force is rapidly reduced because the streamwise velocity component (U) is now parallel to the angular velocity $\Omega$. But the Corilis force still can be produced by crossstream component (W), which results in small vortex near the trailing-outer surface. As the fluid enters the U-turn the colder heavier fluid near the trailing surface is accelerated first then the lighter hotter fluid next. This causes Prandtl’s secondary flow of the second type, (stress-induced) due to the anisotropy of the turbulent Reynolds stresses, to appear in the leading-outer corner for Ro=0.43. Increasing rotation number, the corner vortices are suppressed and the cross-stream secondary motion is governed by the Coriolis force and pressure gradient where a third small vortex appeared near the trailing-outer surfaces at Ro=1.29. Increasing DR did not affect the structure of the vortices much. At the U-turn exit (plane Q6) three effects are interacting: the Corilis force that pushes the cold fluid towards the leading surface, the effect of the circulation in the center of the U-bend and the buoyancy force aligned with meanstream flow. The combined effect of curvature, Coriolis and centrifugal buoyancy forces resulted in appearance of three vortices (at Ro=0.43): large vortex near the leading, a vortex near the trailing, and a small vortex near the leading-outer sides. Increasing Ro up to 1.29, the Coriolis induced large vortex dominates, which is shifted downward, and the two small vortices are suppressed. Increasing DR to 0.229, a small vortex appears at the trailing-inner corner and a second vortex appears at the leading–inner surface as DR increased to 0.4 as a result of increasing buoyancy.
Figure 6. 20 Secondary Flow Vectors
6.3. 4 The Influence of Fij on Dynamics

The system rotation at angular velocity $\Omega^*$ is manifested in the velocity fluctuation equation by the appearance of a Coriolis force term, $-2(\Omega^* \times V)$. The additional forces in orthogonal-mode rotation that are acting on the flow are:

**Coriolis force**: $f_{co} = -2\rho(\Omega \times V)$, which in this case yields: $f_{co} = -2\rho[j(\Omega^w) - k(\Omega^v)]$. In the first pass the Coriolis force due to $W$ velocity component acts toward trailing surface and toward leading surface in second pass and $V$ is much smaller than $W$. In the U-turn the Coriolis force due to $+V$ and $-V$ velocity components acts towards outer and inner surfaces, respectively.

**Centrifugal force**: $f_{ce} = -\rho[\Omega \times (\Omega \times r)]$, which yields: $f_{ce} = k(\rho \Omega^2 r)$. In the first, second passes and in the U-turn the centrifugal force acts toward positive $z$ direction away from axis of rotation.

The derived equation for the production term by system rotation is given by:

$$F_{ij} = -2\rho \Omega^k (\overline{u_j u_m \varepsilon_{ikm}} + \overline{u_i u_m \varepsilon_{jkm}})$$ (6.1)

In case of orthogonal mode rotation the influence of Eq. 6.1 on the dynamics is to be investigated next. With a rotation vector $\Omega^* = (-\Omega i + 0j + 0k)$, it follows from Eq. 6.1 that the Reynolds stresses equations contain additional terms as shown in Table 6.1:
Table 6. 1 The Influence of Fij on Dynamics in Orthogonal-Mode Rotation

<table>
<thead>
<tr>
<th>i,j</th>
<th>Re Stress</th>
<th>Fij</th>
<th>i,j</th>
<th>Re Stress</th>
<th>Fij</th>
<th>i,j</th>
<th>Re Stress</th>
<th>Fij</th>
</tr>
</thead>
<tbody>
<tr>
<td>3,3</td>
<td>w w</td>
<td>4(\rho \Omega w w)</td>
<td>1,1</td>
<td>u u</td>
<td>0</td>
<td>3,3</td>
<td>w w</td>
<td>4(\rho \Omega w w)</td>
</tr>
<tr>
<td>2,2</td>
<td>v v</td>
<td>-4(\rho \Omega v w)</td>
<td>2,2</td>
<td>v v</td>
<td>-4(\rho \Omega v w)</td>
<td>2,2</td>
<td>v v</td>
<td>-4(\rho \Omega v w)</td>
</tr>
<tr>
<td>3,2</td>
<td>w v</td>
<td>2(\rho \Omega (v^2 - w^2))</td>
<td>1,2</td>
<td>u v</td>
<td>-2(\rho \Omega u w)</td>
<td>3,2</td>
<td>w v</td>
<td>2(\rho \Omega (v^2 - w^2))</td>
</tr>
</tbody>
</table>

Since, nominally, \(w w > v v\) in channel flow, thus in the first pass F32 is negative, in which case it acts to add to Reynolds shear stress on the trailing surface where \(w v < 0\) and decrease it on the leading surface where \(w v > 0\) if no density ratio is involved. Enhanced Reynolds shear stress may be associated with high turbulence activity and hence the trailing side is unstable (pressure side) and the leading (suction side) is stable. In second pass the opposite is valid.

To study why the Coriolis force has that kind of effect on the Reynolds shear stress, consider the part of F32 that depends on \(w w\). This term has arisen from multiplying the Coriolis term \(-2(\Omega i W)\) by W. The expression \(-2(\Omega i W)\) represents a force deflecting W \(> 0\) motions toward the trailing surface and motions W \(< 0\) toward the leading surface. In other words, W \(> 0\) motions are accompanied by a move negative V, while W \(< 0\) motions make V move positive. Thus it is expected that the Reynolds shear stress is amplified on the trailing side and decreases on the leading side in case of no density ratio variations.

Similarly, the term \(2\rho \Omega v^2\) in F32 acts to diminish the Reynolds shear stress. In this case the Coriolis force \(-2(\Omega i V)\), deflects motion in the \(\pm Y\) directions toward the \(\pm Z\) directions, respectively, which leads to reduce the Reynolds stress. The cumulative effect of the two
processes represented in F32 is to enhance Reynolds stress on the trailing side while decreasing it on the leading side because the V fluctuations are smaller than the W fluctuations and hence the corresponding Coriolis effect is smaller (if no heating is involved).

In the $\bar{w}\bar{w}$ equation there is an extra production term $4\rho \Omega \bar{w}\bar{v}$ and an equal and opposite term in the $\bar{v}\bar{v}$ equation. When $\bar{w}\bar{v}$ is negative means that rotation diminishes $\bar{w}\bar{w}$ and enhances $\bar{v}\bar{v}$ on the trailing side and vice versa on the leading side. For high Ro value considered in this dissertation, it can happen that $\bar{w}\bar{w} < \bar{v}\bar{v}$, in which case, according to F32 equation there is no new effect of rotation on enhancing or diminishing Reynolds stress.

Since F33 in $\bar{w}\bar{w}$ equation appears with equal and opposite magnitude to F22 in the $\bar{v}\bar{v}$ normal Reynolds stress equation, there is clearly no net effect of rotation in the turbulent kinetic energy equation. Here it is worth to note that rotation effects in two-equation models do not appear in the mean momentum equation, except indirectly through the action of the Reynolds shear stress. Consequently, there is no mechanism by which a model such as k-ε model is able to account for the effects of rotation in a channel flow. Clearly, the minimum requirement necessary to do so is to have the capacity to model the anisotropy of the Reynolds stress tensor in unidirectional mean flows. Only more advanced models such as RSM are rotationally useful for treating rotating channel flow. Also a good quality of results depends strongly on the attention paid to the near-wall modeling.

To illustrate the fundamental advantage of the exact formulation of the production terms in RSM compared to EVMs, consider $F_{ij}$ production term. The k-equation ($k = 0.5(\bar{u}\bar{u} + \bar{v}\bar{v} + \bar{w}\bar{w})$ is
unaffected by rotation as F33+F22=0. In addition the shear stress, \( \bar{w} \bar{v} \), is inaccurately estimated by neglecting the rotational production F32 because of the isotropic estimation of the normal stresses \( \bar{w} \bar{w} = \bar{v} \bar{v} \). This clearly suggests that EVMs should include ad hoc modifications to correctly predict rotational induced turbulence.

6.3.4.1 Predicted Reynolds Shear Stress Components

The shear stresses, \( \bar{v} \bar{w} \), are depicted in Figure 6.21 and Figure 6.22. At the entrance of the U-turn (location N4), high rotation (strong Coriolis) and buoyancy numbers cause the increase of shear stresses near both trailing side (where \( V'W' <0 \)) and leading side (where \( V'W' >0 \)). Where an increase by a factor of 4 of that for low rotation numbers was observed at the entrance of the U-turn. At the center of the U-turn (location P5), increasing Ro and DR causes a sharp increase in shear stress near the trailing and convex (inner) surfaces (where \( V'W' >0 \)), decreases them near outer (concave) surface and increases them near leading surface. At the U-turn exit (location Q6) the turbulence has been greatly damped over a large region and the higher shear stress is found near the trailing- concave (outer) surfaces and with change in sign near the leading surfaces and is mainly caused by increasing both DR and Ro. As a general trend, it has been observed that the +ve \( V'W' \) are increasing at high Ro by increasing DR, while for low Ro cases they are decreasing.

The increase in turbulence activity is usually expected to cause an increase in heat transfer rate. The increase in shear stresses due to increasing DR mentioned above is not necessarily leading to
increase in heat transfer rate as the fluid temperature, the wall temperature and the wall heat flux are also important factors that should be considered when observing the heat transfer behavior.

Figure 6. 21 Reynolds Shear Stress Components in Vertical Direction (Between Trailing and Leading Surfaces)
Figure 6. 22 Reynolds Shear Stress Components in Horizontal Direction (Between Inner and Outer Surfaces)
6.3.4. 2 Normal Reynolds Stress Components

An important measure of any turbulent flow is how intense the turbulent fluctuations are. This can be quantified in terms of the specific Reynolds stress components, $u'^2$, $v'^2$, and $w'^2$. These three normal stresses can also be regarded as the kinetic energy per unit mass of fluctuating velocity field in the three coordinate directions. These Reynolds stresses are normalized relative to the flow inlet velocity to give the relative turbulence intensity. The normal stresses, $w'^2$, are depicted in Figure 6.23 and Figure 6.24. It has been observed that at the U-turn entrance normal stresses, $w'^2$, increase by increasing Ro to 1.29 and density ratio to 0.4 by a factor of 6 near the trailing surface and by a factor of 3 near leading surface compared to the case with Ro = 0.43 and DR = 0.13. The increase in turbulent intensities has favorable effect on the heat transfer. At the center of the U-turn (P5) the normal stresses, $u'^2$ are still high but less than at the entrance and the increase in normal stress favors the concave (outer) and leading surfaces. At the U-turn exit the normal stresses, $w'^2$ are much less in magnitude (turbulence decay) and higher intensities were found near the outer surface, while they are of the same magnitude near leading and trailing surfaces. The well-known anisotropy in the normal stresses was observed in all planes and it was shown in previous work by the author, see [141].
Figure 6. 23 Stream Wise Normal Stress Components in Vertical Direction
Figure 6. 24 Stream Wise normal Stress Components in Horizontal Direction
The flow in the U-turn region is strongly affected by pressure gradients and Coriolis force due to \( V \) velocity component. To observe these effects on heat transfer rate the local Nu is studied. Table 6.2 shows the calculated Nusselt number at three locations: inlet, center and exit of the U-turn for different Ro and DRs. The results are summarized as follows:

**Leading surface:** at the inlet of the U-turn location N4, Nusselt number increases by a factor of two by increasing Ro from 0.43 to 1.29 then decreases slightly by increasing DR from 0.13 to 0.4. At the center (Location P5) and the exit (Location Q6) of the U-turn Nusselt number increases slightly and by a factor of 1.7 respectively by increasing Ro then decreases slightly by increasing DR.

**Trailing surface:** at the inlet of the U-turn location N4, Nusselt number increases by increasing Ro then decreases slightly by increasing DR. At the center (Location P5) Nusselt number increases by a factor of 2.4 by increasing Ro, while increasing density ratio decreases Nusselt number. At the exit (Location Q6) of the U-turn Nusselt number increases slightly by increasing Ro and DR.

**Inner surface:** at the inlet of the U-turn (location N4), Nusselt number increases by a factor of 1.6 by increasing Ro then decreases slightly by increasing DR. At the center (Location P5) and the exit (Location Q6) of the U-turn Nusselt number increases slightly by increasing Ro then decreases slightly by increasing DR. This behavior is the same as on the leading surface.
Outer surface: at the inlet of the U-turn (location N4), at the center (Location P5), Nusselt number increases by increasing Ro then decreases slightly by increasing DR. At the exit (Location Q6) of the U-turn Nusselt number decreases by increasing Ro then increases slightly by increasing DR.

6.3. 6 Average Nusselt Number in the U-turn region

The average Nusselt number versus Ro for four surfaces (leading, trailing, outer and inner) is calculated for the three locations and given in Figure 6. 25 for DR=0.13. The 4-side-average-Nu was calculated based on the bulk temperature at every location. It is clearly shown that Nu correlate with Ro and a linear correlation for Nu as a function of Ro is possible to derive at every level of DR. The increase in average Nusselt number is more in the center than at the entrance and the exit of the U-turn.

Table 6. 2 Local Nusselt Number in the U-turn Region

<table>
<thead>
<tr>
<th>Location</th>
<th>Surface</th>
<th>Ro=0.43 DR=0.13</th>
<th>Ro=0.86 DR=0.13</th>
<th>Ro=1.29 DR=0.13</th>
<th>Ro=1.29 DR=0.25</th>
<th>Ro=1.29 DR=0.4</th>
</tr>
</thead>
<tbody>
<tr>
<td>N4</td>
<td>Leading Y/Dh=1</td>
<td>1.133</td>
<td>1.684</td>
<td>2.310</td>
<td>2.338</td>
<td>2.147</td>
</tr>
<tr>
<td></td>
<td>Trailing Y/Dh=0</td>
<td>2.702</td>
<td>2.566</td>
<td>3.698</td>
<td>3.386</td>
<td>2.823</td>
</tr>
<tr>
<td></td>
<td>Inner X/Dh=3.5</td>
<td>2.170</td>
<td>2.557</td>
<td>3.555</td>
<td>3.251</td>
<td>2.817</td>
</tr>
<tr>
<td></td>
<td>Outer X/Dh=4.5</td>
<td>2.153</td>
<td>2.507</td>
<td>3.423</td>
<td>3.102</td>
<td>2.716</td>
</tr>
<tr>
<td></td>
<td>Trailing Y/Dh=0</td>
<td>1.129</td>
<td>2.064</td>
<td>2.663</td>
<td>2.768</td>
<td>2.691</td>
</tr>
<tr>
<td></td>
<td>Inner Z/Dh=15.25</td>
<td>2.937</td>
<td>3.131</td>
<td>3.699</td>
<td>3.105</td>
<td>2.447</td>
</tr>
<tr>
<td>Q6</td>
<td>Leading Y/Dh=1</td>
<td>1.349</td>
<td>1.818</td>
<td>2.289</td>
<td>2.082</td>
<td>1.901</td>
</tr>
<tr>
<td></td>
<td>Trailing Y/Dh=0</td>
<td>1.890</td>
<td>2.103</td>
<td>1.983</td>
<td>2.095</td>
<td>2.129</td>
</tr>
<tr>
<td></td>
<td>Inner X/Dh=1</td>
<td>2.215</td>
<td>2.275</td>
<td>2.831</td>
<td>2.532</td>
<td>2.167</td>
</tr>
<tr>
<td></td>
<td>Outer X/Dh=0</td>
<td>2.342</td>
<td>1.854</td>
<td>1.763</td>
<td>2.118</td>
<td>2.163</td>
</tr>
</tbody>
</table>
6.3.7 Total Pressure Drop

The total pressure drop is calculated according to Equation 6.2:

\[
\left[ P_1 + \frac{1}{2} \alpha \rho_1 W_1^2 \right] - \left[ P_s + \frac{1}{2} \alpha \rho_s W_s^2 \right] = \omega^2 (\rho_s R_s - \rho_1 R_1) + \text{Losses} \tag{6.2}
\]

where the subscript 1 and S correspond to inlet and location in the stream wise direction, respectively.

Figure 6.26 shows the mass weighted average pressure drop across the channel. In the first pass the total pressure increases as Ro increases to reach the maximum value in the U-turn, then starts to decrease in the second pass. Increasing DR for the same Ro decreases the total pressure as the density of the fluid is decreasing. When DR increases to 0.23 and 0.4 at the exit of the channel,
where \( R_S = R_1 \) but \( \rho_S \) is much less than \( \rho_1 \), which means that \( \omega^2 (\rho_S R_S - \rho_1 R_1) \) is less than when \( DR=0.13 \) and hence the pressure gain is more.

![Graph of pressure drop vs S/Dh](image)

**Figure 6. 26 Total Pressure Drop-Smooth Walls Channel**

6. 4 Effect of Coriolis and Centrifugal Forces on Turbulence and Heat Transfer at High Rotation Numbers and Density Ratios in a Rib-Roughened Internal Cooling Channel

Study was performed using the rib roughened channel of Wagner et al. [85] but with rotation numbers of 0, 0.475 and 1 at 0.13 DR and for \( Ro=1 \) at \( DR=0.13, 0.23 \) and 0.3. The Reynolds number and radius ratio \( R/Dh \) were held constant at 25,000 and 44, respectively. The results from this study were also published, see [143].
6.4.1 Temperature Distribution, Streamwise Velocity and Secondary Flows

Figure 6.27 shows the streamwise velocity vectors in a vertical cross section at the center of the 1st pass, U-turn and 2nd pass respectively. Figure 6.28 and Figure 6.29 show the secondary flow patterns, where each figure is viewed from upstream. In the first pass, for Ro=0, the Reynolds stress anisotropy generate secondary flow at the corners and a small circulation bubble is formed in front and after each circular rib on both leading and trailing surfaces. Increasing Ro to 0.475 caused the formation of large thin vortices between ribs in the leading surface as a result of the effect of the ribs and the buoyancy force that decelerate the slow momentum hotter fluid near the leading surface. The centrifugal force accelerated the colder fluid near the trailing surface where the small vortices become clearer. At this level of rotation (Ro=0.475) two Coriolis induced vortices appear near the trailing surface at location S/Dh=8.5 and two more vortices appear at slightly higher location. At Ro=1 the main flow completely gets separated near the leading side. Coriolis and buoyancy/centrifugal effects are much stronger causing the small vortices near the leading surface to become large vortices in the streamwise direction and the small vortices at the trailing rib-back to become wider and those at the trailing rib-front shorter. Increasing DR to 0.23 tends to decrease the size of vortices near the leading side. In the U-turn the flow separates at Ro=1 and DR=0.13 and 0.23. Two vortices are formed in the U-turn, (S/Dh=16.75), as a results of the combined effect of Cross-stream Coriolis forces and high-pressure gradients. In the second pass, where the Coriolis is acting toward the leading surface and buoyancy aligned with streamwise flow, high rotation (at Ro=1) accelerates the flow and tends to suppress the vortices in front of the ribs. Increasing DR to 0.23 causes more flow acceleration and tends to suppress the vortices behind the ribs.
Figure 6. 27 Streamwise Velocity Vectors (W and U) and Temperature (θ) Contours in First Pass (Left), U-turn (Center) and Second Pass (Right)
Figure 6. 28 Secondary Flow Vectors and Dimensionless Temperature (θ) Contours
6.4. 2 Turbulence Anisotropy

The complexity of the flow inside internal cooling channels (ribs, U-turn, rotation etc.) generates turbulence anisotropy. Figure 6. 30 depicts the ratio of the Reynolds stress normal components in the first pass, U-turn, and second pass. In the first pass, where the Coriolis force is directed
toward the trailing surface such that the streamwise velocity profiles are skewed toward the trailing wall, the rib generated shear layer on the trailing side and the high velocity gradients associated with skewed velocity profiles caused high turbulence anisotropy, approximately having values of 0.4-1.9 at $Ro=0.475$ and $DR=0.13$, 0.3-1.6 for $Ro=1$ and $DR=0.13$ and of 0.32-1.9 for $Ro=1$ and $DR=0.23$ near the leading surface. In the second pass, the turbulence anisotropy has values of 0.4-1.8 at $Ro=0.475$ and $DR=0.13$, 0.4-1.8 for $Ro=1$ and $DR=0.13$ and of 0.27-1.7 for $Ro=1$ and $DR=0.23$ near the trailing surface. In the U-turn, the turbulence anisotropy has values of 0.55-1.8 at $Ro=0.475$, 0.5-1.7 for $Ro=1$, $DR=0.13$ and of 0.4-1.8 for $Ro=1$, $DR=0.23$ near the trailing surface. As a comparison, Liou et al. [95] in their experimental study of flow and heat transfer in a rotating two-pass square channel with 90 deg in-line square ribs on leading and trailing surfaces reported turbulence anisotropy, also in terms of the ratio of the Reynolds stress normal components, of 1.8-2.2, 1.8-2.8 and 1.3-2.4 in the first pass, U-turn, and second pass, respectively for $Ro=0.2$. The results in this study show that high rotation and density ratios, although suppresses turbulence in general trend but it also tend to redistribute these intensities resulting on large low intensities and small high intensities regions.
Figure 6. 30 Turbulence Anisotropy in Terms of $W'W'/V'V'$ Contours in the First and Second Pass and in Terms of $U'U'/V'V'$ in the U-turn

6.4. 3 Reynolds Shear Stress Components

The shear stresses, $\overline{vw}$, play a dominant role in the theory of mean momentum transfer by turbulent motion. They are depicted in Figure 6. 31 and Figure 6. 32 (normalized with $W_0^2$) in the first pass, U-turn and second pass. In the first pass, high rotation (strong Coriolis), and buoyancy increased shear stresses near the trailing surface (with minus sign) and unlike stationary and low Ro cases, also increases them near the leading surface (with positive sign). In the U-turn at Ro=1, two regions are predicted with high shear stress; 0.02 near the leading and –0.08 near the trailing surfaces. Increasing density ratio to 0.23 caused an increase of $\overline{vw}$ to 0.065 near leading and –0.108 near trailing, respectively. In the second pass shear stresses increased near the leading surface (with –ve sign) and also increased near the trailing surface (with +ve
sign) by increasing rotation number to 1. Increasing DR to 0.23 caused further increase in shear stresses near both leading and trailing surfaces.

Figure 6. 31 Reynolds Shear Stresses (Z/Dh=0 Corresponds to Inner Surface in the U-Turn)
Figure 6. 32 Reynolds Shear Stresses in Terms of $V'W'/W_o^2$ Contours in the First and Second Pass and in Terms of $U'V'/W_o^2$ in the U-turn. (Values Shown are the +ve and –ve maximum)

6.4. 4 Nusselt Number

In order to study the effect of increasing rotation number and density ratio on heat transfer distribution, Nusselt number (normalized with Nusselt number at Ro=0 and DR=0.13) was calculated at 9 locations on the leading, trailing, inner and outer surfaces for Ro of up to 1 and DR of up to 0.3. Calculations of Nusselt number were based on the mass weighted average bulk temperature at every section of the 9 locations. The results are shown in Figure 6. 33 and Figure 6. 34. Note that the heat transfer coefficients in Wagner et al. [85] were based on the projected area rather than the actual heat transfer surface area due to the rib geometry, where the actual heat transfer area is 1.15 times the projected area. On the leading surface, for every level of rotation, Nu decreases up to $S/D_h=13$ then starts to increase up to $S/D_h=17$, then Nu decreases
up to $S/D_h=19$, after that $Nu$ increases all the way to the exit of the channel. Increasing DR to 0.23 increased $Nu_r$ on the first and second passes and on the U-turn region. Increasing DR to 0.3 increases $Nu_r$ slightly on the first pass and decreased it slightly on the U-turn and second pass compared to the case of DR=0.23.

Figure 6. 33 Effect of Increasing $Ro$ and DR on Local $Nu/Nuo$ on Leading and Trailing Surfaces
Figure 6. Effect of Increasing Ro and DR on Local Nu/Nuo on Inner and Outer Surfaces

On the trailing surface for every level of Ro Nur increases rapidly up to S/Dh = 8.5 then starts to decrease up to S/Dh=12.4 in the first pass. The increase in Nur due to increasing Ro is much more than in the U-turn and second pass. Increasing DR reduces Nur (compared to the case with DR=0.13) in the first and second passes and slightly increases Nur in the U-turn. On the inner surface Nur increased for every Ro up to S/Dh=10 then decreased slightly then picked up its maximum value on the U-turn region after which it decreases to its minimum values. Increasing Ro caused an increase on Nur while increasing DR decreased Nur for all location on the inner surface. On the outer surface, increasing Ro increases Nur in all locations, while increasing DR
decreases Nur. The maximum increase in Nur was found on the first pass and the U-turn region, while the maximum decrease was found on the second pass. The decrease in Nur as DR increases is due to the increase in thermal boundary layer near the walls, which is blocking the colder fluid from flowing very close to the channel walls. From these results it has been observed that the 4-side-average Nur increases linearly with Ro. Thus it is possible to derive correlation for Nur/Nus as a function of rotation number and another correlation as a function of density ratio for every surface (i.e. leading, trailing, inner and outer) or for the whole channel. Such correlation for the whole channel was derived based on Ro of 0.24, 0.35, 0.475, 0.74 and 1 for DR of 0.13 and it’s given as:

\[
\frac{\text{Nur}}{\text{Nus}} = 0.606 \times \text{Ro} + 1
\]  

(6.3)

The variation of Nur with DR is not significant at high rotation numbers. At low rotation numbers (up to Ro=0.35), increasing density ratio caused an increase in Nu on the leading and trailing surfaces at all locations of the channel. This behavior of Nu is confirmed in the experimental results by Wagner et al. [85] for Ro up to 0.35 and DR up to 0.23.

6.4. 5 The Brunt-Vaisala Frequency

Brunt-Vaisala frequency "N" is a measure (in inverse time units) of the stratification or resistance to turbulence. To derive this concept, imagine displacing a fluid particle vertically in a stably stratified fluid column. If the particle were to be lifted a distance \( \Delta Z \), it would be denser than the surrounding fluid and experience a downward force related to the difference in density between it and the surrounding fluid:
\[ F = -g \Delta \rho = -g \Delta \frac{\partial \rho}{\partial z} \quad (6.4) \]

which in turn would be a function of the distance of displacement and the density gradient. The equation applies for negative displacements as well, since the particle would be more buoyant than the surrounding fluid, and want to "cork" back up. With \( F = ma \), then the equation becomes:

\[ \rho a = -gz \frac{\partial \rho}{\partial z} \quad (6.5) \]

or

\[ \frac{\partial^2 z}{\partial^2 t} = -\frac{g}{\rho} (\frac{\partial \rho}{\partial z}) z \quad (6.6) \]

where the terms in front of the "z" on the right hand side of the equation can be expressed as a constant, \( N^2 \). The solution to this equation is:

\[ z(t) = z_0 e^{iNt} \] a simple harmonic oscillator with frequency \( N \),

\[ N = \sqrt{\frac{g \frac{\partial \rho}{\partial z}}{\rho \frac{\partial \rho}{\partial z}}} \quad (6.7) \]

Substitute \( g = \omega^2 R \), then \( N \) becomes:

\[ N = \sqrt{\frac{\omega^2 R \frac{\partial \rho}{\partial z}}{\rho \frac{\partial \rho}{\partial z}}} \quad (6.8) \]

the Brunt-Vaisala Frequency. That is, if you were to "pluck" a fluid particle above its equilibrium position, it would return to its position, but overshoot, oscillating back and forth with a frequency "N". This frequency is a measure of the stability. Figure 6.35 gives \( N \) in the center line of the first pass and Figure 6.36 in the center line of the second pass of the cooling channel.
Increasing Ro from 0.238 to 1 with the same density ratio of 0.13 causes an increase in N in some locations and decrease in N in other locations of the first pass and causes an increase in N in the second pass. Note that N in the second pass is much less in magnitude than in the first pass. Increasing density ratio to 0.3 at high Ro of 1 causes an increase in N in both first and second passages, which means that the fluid tends to oscillate back and forth with a higher frequency N trying to return to its equilibrium position.

Figure 6. 35 Brunt Vaisala Frequency at the Center of the First Pass. The Flow Direction is from Left to Right
Figure 6.36 Brunt Vaisala Frequency at the Center of the Second Pass. The Flow Direction is from Right to Left

6.4.6 Total Pressure Drop

Figure 6.37 shows the mass weighted average pressure drop across the channel. In the first pass the total pressure increases (with –ve sign) as Ro increases to reach the maximum value in the U-turn, then starts to decrease in the second pass. As with smooth wall channel case, increasing DR increases the total pressure gain (with –ve sign) near the channel exit because the density of the fluid is decreasing.
In case of parallel mode rotation the influence of eq. 6.1 on the dynamics is also investigated. With a rotation vector $\Omega^* = (0i + 0j + \Omega k)$, it follows from eq. 6.1 that the Reynolds stresses equations contain additional terms as shown in Table 6.3. The additional forces in parallel-mode rotation that are acting on the flow are:

Coriolis force: $f_{co} = -2\rho(\Omega \times V)$, which in this case yields: $f_{co} = 2\rho[i(\Omega v) - j(\Omega u)]$, which has two components. The first component acts toward the left surface when $v$ is positive and toward right surface when $v$ is negative. The second component acts toward bottom surface when $u$ is positive and toward top surface when $u$ is negative.
Centrifugal force: \( f_{ce} = -\rho[\Omega \times (\Omega \times r)] \), which yields: \( f_{ce} = -j(\rho \Omega^2 r) \), which acts toward the bottom surface.

Table 6.3 The Influence of Fij on Dynamics in Parallel-Mode Rotation

<table>
<thead>
<tr>
<th>i,j</th>
<th>Reynolds stress</th>
<th>Fij</th>
<th>i,j</th>
<th>Reynolds stress</th>
<th>Fij</th>
</tr>
</thead>
<tbody>
<tr>
<td>1,1</td>
<td>( \frac{u}{u} )</td>
<td>4( \rho \Omega \frac{u}{v} )</td>
<td>1,2</td>
<td>( \frac{u}{v} )</td>
<td>4( \rho \Omega \frac{v}{v} )</td>
</tr>
<tr>
<td>2,2</td>
<td>( \frac{v}{v} )</td>
<td>-4( \rho \Omega \frac{v}{u} )</td>
<td>1,3</td>
<td>( \frac{v}{v} )</td>
<td>2( \rho \Omega \frac{v}{w} )</td>
</tr>
<tr>
<td>3,3</td>
<td>( \frac{w}{w} )</td>
<td>0</td>
<td>3,2</td>
<td>( \frac{w}{v} )</td>
<td>-2( \rho \Omega \frac{w}{u'} )</td>
</tr>
</tbody>
</table>

6.5.1 Average and Local Nusselt Number

The average Nusselt number values from this study were compared to experimental results of Morris 81 for Ro=0 (no rotation) and Ro=0.063 (the highest experimental Ro) for wall heat flux of q"1=6.45 kw/m². The comparison is shown in Figure 6.38. The average Nu number from this study was taken as the average of the four sides of the channel, while in Morris 81 the local Nusselt number is provided and it is not clear whether the difference in Nu between the four surfaces of the channel is considered. Moreover, the inlet and exit boundary conditions assumed in this study were different from the experimental boundary conditions, which are not known. However Figure 6.38 shows that there is a difference of up to 22% in measured and calculated Nu. This difference is believed to be due to the difference in boundary conditions and may be due to difference between the experimental and predicted channel geometry.
The calculated local Nusselt number on the left surface of the channel is shown in Figure 6. 39 At the entrance of the channel the flow separates as Ro increases, which results in Nu number increase up to $Z/L=0.1$, then sharp decrease up to $Z/L=0.2$. After that Nu maintains nearly constant value up to the exit of the duct. Increasing Ro causes an enhancement in heat transfer due to the effect of Coriolis force that acts toward left surface, while increasing wall heat flux at high rotation number of 0.35 decreases Nu to same values as for the stationary case.

On the right surface (Figure 6. 40) almost the same behavior of Nu is observed as on the left surface.
Figure 6. 39 Nusselt Number Distribution on the Left Surface

Figure 6. 40 Nusselt number Distribution on the Right Surface
The highest values of Nu are found on the bottom surface, (Figure 6. 41) toward which the Coriolis and centrifugal forces are acting. Increasing Ro to 0.14 causes Nu to increase significantly, while further increase of Ro to 0.35 resulted in less increase in Nu beyond Z/L =0.5. The explanation for this could be as follows: for Ro=0.14 the effect of Coriolis force that acts toward the bottom surface adds to the effect of centrifugal force, which also acts toward the bottom surface, which increases Nu rapidly. Increasing Ro to 0.35 the centrifugal force increases while it is quite possible that the Coriolis force changes its direction to act toward the top surface. The net effect of the two forces is still to increase Nu, but with less rate than for Ro=0.14. Increasing wall heat flux at Ro=0.35 decreases Nu slightly because the thermal boundary layer near the bottom surface increases.
On the top surface, (Figure 6. 42) Nu increases for Z/L<0.2 as Ro and wall heat flux increase then decreases rapidly for Z/L>0.2. The strong centrifugal force pushes the colder fluid away from the top surface and thus the fluid near this surface is always hotter, which yields lower Nu.

The decrease in Nu on all surfaces as the wall heat flux increases could be associated with the fact that the difference between the wall and the fluid temperatures increases, which means that the heat transfer coefficient decreases and consequently Nu decreases.

These results of local Nu on all four surfaces of the channel clearly show that due to effect of centrifugal and buoyancy forces the heat transfer rate differs substantially from each surface especially from top and bottom surfaces. The correlation for Nu given by Equation 4.1 in chapter 4 fails completely in such situation. This suggests that new correlations as a function of Ro and
buoyancy parameter are needed to describe the heat transfer rate. This correlation should take into account both low and high Ro and buoyancy parameter.

6.5. 2 Secondary Flow

The secondary flow vectors at three axial locations are shown in Figure 6.43. At Z/L=0.3, for low Ro of 0.063 and 0.14 two vortices are formed near the left-top and right-bottom surfaces. At this location for low Ro the combined effect of Coriolis and centrifugal forces caused the appearance of these two vortices. Further downstream the centrifugal force effect dominates, which resulted in two large vortices moving toward the bottom surface. The colder fluid is pushed toward the bottom surface, and then flows back near the left and right surfaces, which caused an increase in Nu near these surfaces.

Increasing Ro to 0.35 increases the centrifugal force, which has more effect than the Coriolis force resulting in two large vortices at Z/L=0.3. The height of these vortices decreases downstream at locations Z/L=0.6 and 0.9 due to the increase in wall temperature of the top surface.

Increasing wall heat flux at Ro=0.35 did not change the structure of the vortices much.
Figure 6. 43 Secondary Flow Vectors Colored by Streamwise Velocity
6.5. 3 Reynolds Shear Stress Components

Reynolds shear stress components are depicted in Figure 6.44 at location Z/L=0.8 and in Figure 6.45 at a location in between Z/L=0.8 and Z/L=0.9. According to Table 6.3, when V’W’ has the same sign as W’U’, then W’U’ acts as a sink to reduce V’W’. While when V’W’ has different sign from W’U’ then W’U’ acts as a source to add to V’W’. Figure 6.44 and Figure 6.45 show that W’U’ always has the same sign as V’W’, hence W’U’ acts as a sink to reduce V’W’. As Ro and wall heat flux increase, W’U’ increases near both top and bottom surfaces, while V’W’ increases near bottom surface (yielding to increase Nu) and decreases with rotation (yielding to decrease Nu) then increases as wall heat flux increases near top surface. The increase of the V’W’ shear stress near the bottom surface enhances the heat transfer rate, while the decrease of V’W’ near the bottom surface decreases the heat transfer rate. As for the orthogonal mode rotation, the slight increase in V’W’ due to increasing wall heat flux does not enhance heat transfer rate because the thermal boundary layer becomes thicker.
Figure 6. Reynolds Shear Stress Components (m²/s²) at Z/L=0.8 (Y/Dh=0 corresponds to bottom surface and Y/Dh=1 corresponds to top surface)
Figure 6. 45 Reynolds Shear Stress (m²/s²) Components in Between Z/L=0.8 and Z/L=0.9

6.5. 4 Reynolds Normal Stress Components and Turbulence Anisotropy

Figure 6. 46 show the normal stress components W’W’ and V’V’ AT Z/L = 0.8. Normal stresses increase near bottom surface as Ro and wall heat flux increase and decrease near top surface as Ro increases and then increase as wall heat flux increase. Figure 6. 47 shows the turbulence anisotropy contours in terms of W’W’/V’V’. At Ro=0.063 the flow is isotropic in the center of the channel. Increasing Ro to 0.14 and 0.35 the flow becomes anisotropic. Increasing wall heat flux to q”2 reduced W’W’/V’V’, while further increase of wall heat flux to q”3 increases W’W’/V’V’ slightly.
Rotation has no effect on $W'W'$ if no heating is involved, see Table 6.3, and thus the key component here is $V'V'$.

Figure 6. 46 Reynolds Normal Stress Components ($m^2/s^2$) at $Z/L=0.8$ ($Y/Dh=0$ corresponds to bottom surface and $Y/Dh=1$ corresponds to top surface)
Figure 6.47 Turbulence Anisotropy in Terms of $W'W'/V'V'$ Contours in Between $Z/L=0.8$ and $Z/L=0.9$

6.5.5 Mass Weighted Average Total Pressure Drop

The mass weighted average total pressure drop along the channel is calculated relative to inlet location ($P_1$). The results are shown in Figure 6.48. The total pressure drop is increasing almost linearly with increasing $Ro$ and also increasing slightly with increasing wall heat flux because of the increasing effects of secondary flows, mixing and friction.
Figure 6. 48 Mass Weighted Average Total Pressure Drop – Parallel Mode Rotation
CHAPTER SEVEN: CONCLUSIONS

Comparing two-equation turbulence models and RSM, the following is concluded:

The two-equation turbulence models predicted incorrect trends when applied to non-equilibrium, anisotropic, three-dimensional, with U-turn, secondary flows, with separation and with rotating and stratified coolant flow studied in this dissertation.

The two-equation models need a length scale correction and corrections to account for streamline curvature, buoyancy and rotation.

The standard wall functions do not yield accurate results when applied under conditions different from those under which the law of the wall is derived. The velocity profile near the wall is altered by the pressure gradients, buoyancy fluxes and non-equilibrium flow.

RSM (with enhanced wall treatment that resolves the sublayer region) accounts for history effects, as the Reynolds stress equation includes convection and diffusion terms for the stresses and eliminate the need for equilibrium assumption and local isotropy. Also, the Reynolds stress equations include production and body force terms that can respond to the effects of the streamline curvature, rotation and buoyancy. Hence, RSM predictions are superior to the two-equation models predictions.
Studying the flow field and heat transfer in rotating internal cooling channels with smooth walls at high rotation and buoyancy numbers, the following is concluded:

On the leading side of the first pass Nusselt number ratio increases rapidly for high rotation numbers then decreases (to almost the same values as for low rotation numbers) before and in the U-turn and continued to decrease up to the exit. In the trailing side of the first pass high rotation number has increasing effect on Nusselt number ratio for S/D of up to 7 then it starts to decrease.

Increasing Ro always increases Nu, while at high Ro, increasing DR dose not increase Nu and in some locations even decreases Nu. The increasing thermal boundary layer thickness near walls is the possible reason for this behavior of Nu.

In the first pass the combined effect of the Coriolis and centrifugal forces causes the separation near the leading surface and the acceleration of the flow near the trailing surface, which led to the rapid increase in Nusselt number. In the second pass the stream wise velocity of the well-mixed fluid increases rapidly at high rotation and buoyancy numbers near the leading side and decreases to low values near the trailing surface. The opposite was observed at low rotation numbers.

High rotation and buoyancy numbers has strong effect on Reynolds shear stress, where an increase of up to 4 times of that for low rotation numbers was observed in the first pass. Normal stresses, $\overline{w^2}$, increase by increasing rotation and buoyancy numbers to a very high values in the first pass. In the second pass the normal stresses, $\overline{w^2}$ are much less in magnitude and have higher
values near the leading side. In the U-turn the normal stresses, $\overline{u^2}$ are still high but less than in the first pass. The well-known anisotropy in the normal stresses was observed in all planes.

*The results of a 3-D numerical simulation of the flow field and heat transfer in the U-bend of smooth internal cooling channel for high rotation numbers (up to Ro=1.29) and high-density ratio (up to 0.4) show the following:*

At the center of the U-bend, increasing rotation number, the corner vortices are suppressed and the cross-stream secondary motion is governed by the weak cross-stream Coriolis force and pressure gradient, where a third small vortex appeared near the trailing-outer surfaces at Ro=1.29. Increasing DR did not affect the structure of the vortices. At the U-turn exit, the combined effect of curvature, Coriolis and centrifugal buoyancy forces resulted in the appearance of three vortices (at Ro=0.43). Increasing Ro, the large and small vortices are suppressed, while increasing DR to 0.4, two small vortices appear at the trailing-inner and leading-inner surfaces.

At the U-turn entrance, high Ro increases the rapid acceleration of the colder fluid near the trailing surface (increase in Nu) and the separation of the hotter fluid near the leading surface (increase in Nu), increasing DR causes a decrease in Nu in all surfaces except in trailing surface. At the center of the channel the flow separates and a sever pressure gradient is encountered. Increasing Ro and DR caused rapid increase in shear stress and turbulence intensities mainly caused by increasing both Ro and DR. It is possible to derive linear correlation for the increase in Nu as a function of Ro.

The heat transfer rate can be enhanced rapidly by increasing Ro to values that are comparable to the enhancement due to the introduction of ribs inside internal cooling channels.
Predictions of heat transfer and flow field of a rib-roughened internal cooling channel of turbine blades rotating at high rotation numbers (up to 1) and density ratios (up to 0.3) show the following:

At Ro=1 the flow separates near trailing surface in between ribs and large vortices are formed near the leading surface in the stream wise direction. Increasing DR to 0.23 tends to decrease the size of vortices. In the U-turn the flow separates at Ro=1 and DR=0.13 and 0.23 and two new vortices are formed.

The rib generated shear layer on the trailing side and the high velocity gradients associated with skewed velocity profiles caused high turbulence anisotropy.

In the first pass, high rotation (strong Coriolis), and buoyancy increased shear stresses rapidly near both trailing and leading surfaces. In the U-turn, and second pass increasing Ro and density ratio to 0.23 also caused an increase in shear stresses near leading and trailing surfaces.

Increasing rotation number enhances heat transfer, while increasing density ratio to 0.3 at high Ro did not or in some locations decreases Nur slightly. 4-side-average Nur correlates with Ro linearly, where a linear correlation for Nur/Nus as a function of rotation number is derived.

Studying square internal cooling channels in parallel mode rotation, the following is concluded:

Increasing Ro enhances the heat transfer on three surfaces of the square channel and decreases Nu in the fourth surface. The average Nu increases with Ro. The available in literature
correlations failed completely when applied to high Ro and DR cases considered in this dissertation.

At high Ro, increasing DR dose not increase Nu further, while the total pressure drop increases rapidly.
CHAPTER EIGHT: FUTURE WORK

8.1 Computational Future Work

The effect of high rotation number and high density ratio on turbulence and transport in internal cooling channels that has been discussed so far was performed numerically considering square cross-section channels with smooth and rib-roughened walls maintained at constant temperature. In actual turbine blades and generator rotors the cross-section of internal cooling channel can be rectangular, triangular or circular. Moreover, the surface-heating condition is neither constant temperature nor constant heat flux. The angled ribs as opposed to orthogonal ribs are proved to enhance heat transfer as reported by many research groups at low rotation numbers and density ratios. The internal cooling channel orientation with respect to the rotation direction is of great importance and should be considered when studying internal cooling. Impingment cooling is usually used to cool the leading edge of turbine blades and film cooling holes have important effect on internal cooling. Thus it is important to consider these effects when performing studies on internal cooling. In order for this investigation to be complete and to provide a full understanding and useful set of data for designer it is recommended that the following be studied in the future:

- Effect of Surface-Heating Condition in Both Smooth and Rib-Roughened Channels
• Effect of Rotation on Heat Transfer for Channels with Angled (Skewed) Ribs
• Effect of Channel Orientation with Respect to the Rotation Direction on Both Smooth and Ribbed Channels
• Effect of Channel Cross Section
• Effect of High Rotation Number on Leading-Edge Impingement Cooling
• Effect of Film Cooling Holes

8.2 Experimental Future Work

Performing numerical simulations to study complex physical problems of engineering interest is the only tool when experimental data is not available. But still, for numerical results to be valid, experimental verification is required. Looking to the existing literature it is clear that there is still many aspects unexplored in the cooling of generator rotors and turbine rotor blades, which may possibly help in larger understanding of the subject. In order to provide tools for numerical validation and to bridge the existing gap in knowledge to an extent it is proposed to perform a comprehensive experimental study to see effects of:

- Micro-scale turbulence structures,
- Centrifugal and buoyancy forces,
- Density of the coolant fluid,
- Flow compressibility
- Channel surface roughness
on both orthogonal and parallel cooling duct positions. In order to achieve this objective a test rig has been designed as discussed below.

8.2. 1 Experimental set-up and program

The proposed test set-up is made up of a rectangular cross-section metallic coolant flow passage in a rectangular form simulating the parallel and orthogonal cooling duct as shown in Figure 8.1. Note that in Figure 8.1 the parallel mode rotation test section is shown. The test set-up is designed such that both parallel and orthogonal rotation modes will be assembled interchangeably. Removing the parallel mode rotation duct and installing the orthogonal mode rotation U-duct and vice versa requires the removal and installation of the ballast repeatedly. The cooling duct will be joined on a shaft having concentric flow passage at its both ends to allow coolant flow supply from a pump/compressor. An LN2 unit will be used to keep the temperature of the coolant to a desired low value. The coolant passage will be supplied with coolant fluid in a closed loop system as indicated in the Figure 8.1. The shaft will be coupled with an electric motor with the help of transmission gear system in order to get appropriate rotational speed for the experiments, more than 10,000 RPM in the present case. The cooling duct outer wall will be heated at a constant temperature using heating foils (with insulation on outer side) and a thermostat. Power supply to the heaters will be provided via slip ring mounted on the shaft. The slip ring will also supply power to the anemometers and stepper motors for the measurement purposes. A cross-sensor probe has been planned to take flow measurements, using two TSI 1750A model anemometers. The anemometers will be mounted on the cooling channel arm and
the x-probe will be traversed using the stepper motor. The rest of the electronics will be clamped at the opposite side of the rotating arm, which in part will be helpful in balancing also. At each position of the anemometers the rotating arm will be balanced. Measurements will be taken on at predefined locations on all three arms of the test set-up. Data transmission is planned using a wireless transmission system, which will be linked with a PC to process and store them. The schematic of the proposed data acquisition and anemometer system is shown in Figure 8.2.

The measurements will record the following:

- Air mass flow rate at the inlet
- Air temperature at the inlet and the outlet
- Wall temperature at multiple locations
- Pressure at multiple locations
- $u', v', w'$, hot wire discrete locations with 1-d stepper.
Figure 8.1 Schematic Layout of Proposed Experimental Set-up

Figure 8.2 Schematic for Data Acquisition System
LIST OF REFERENCES


LIST OF PUBLICATIONS

This thesis is based on the work contained in the following publications:


Other relevant publications not included in this dissertation:
