Design of Internal Cooling Passages: Investigation of Thermal Performance of Serpentine Passages

Waseem Siddique

Doctoral Thesis 2011

Division of Heat and Power Technology
Department of Energy Technology
Royal Institute of Technology
Stockholm, Sweden
ABSTRACT

Gas turbines are used to convert thermal energy into mechanical energy. The thermal efficiency of the gas turbine is directly related to the turbine inlet temperature. The combustion and turbine technology has improved to such an extent that the operating temperature in the turbine inlet is higher than the melting temperature of the turbine material. Different techniques are used to cope with this problem. One of the most commonly used methods is internal cooling of the turbine blades. Conventionally air from the compressor is used for this purpose but due to higher heat capacity, steam can be used as coolant. This opens up the possibility to increase the gas temperature. In the case of a combined cycle power plant, its availability provides a good opportunity to be used as a coolant.

The trailing edge of the gas turbine blades is an important region as it affects the aerodynamics of the flow. The aerodynamics demands a sharp and thin trailing edge to reduce profile losses. The conventional method is the release of a lot of cooling air though a slot along the airfoil trailing edge. However in the case of internal only cooling designs, the coolant is not allowed to leave the channel except from the root section to avoid mixing of the gas in the main flow path with the coolant and loss of cooling medium.

The challenge is to design an inner cooling channel, with the cooling medium entering and leaving the blade at the root section, which reduces the metal temperatures to the required values without an increase of the profile losses and at acceptable cooling flow rate and pressure drop.

This thesis presents Computational Fluid Dynamic (CFD) based numerical work concentrated firstly on the flow and heat transfer in two-pass rectangular channels with and without turbulator ribs. The aspect ratio of the inlet pass was reduced to accommodate more channels in the blade profile in chord-wise direction. Additionally, the divider-to-tip wall distance was varied for these channels. Their effect on heat transfer and pressure drop was studied for smooth as well as ribbed channels. It was followed by a numerical heat transfer study in the trapezoidal channel. Different RANS based turbulence models were used to compare the numerical results with the experimental results. Further, new designs to enhance heat transfer in the channel’s side walls (named as trailing edge wall) were studied. These include the provision of ribs at the trailing edge wall only, inline arrangement of ribs at the bottom as well as at the trailing edge wall and a staggered arrangement of these ribs. The final study was a conjugate heat transfer problem with an aim to propose the best internal cooling channel design to reduce the metal temperature of the trailing edge surface for the given thermal and flow conditions. A number of different options were studied and changes were made to get the best possible channel design.

The results show that for a two-pass rectangular channel (both smooth and ribbed), the reduction in inlet channel aspect ratio reduces the pressure drop. For a smooth channel the reduction in the width of the inlet pass does not affect the heat transfer enhancement at the inlet pass and outlet
pass regions. In case of ribbed channels, heat transfer decreases at the tip and bend bottom with decrease in the width of the inlet pass. Among different turbulence models used to validate numerical results against experimental results for case of trapezoidal channel, the low-Re k-epsilon model is found to be the most appropriate. Using the turbulence model that yields results that are closest to the experimental data, the staggered arrangement of ribs at the trailing edge wall is found to have maximum thermal performance. The results from the conjugate heat transfer problem suggest using steam as coolant if it is available as it requires less mass flow rate to get similar wall temperature values as compared to air at similar thermal and flow conditions. It is also found that staggered arrangement of ribs is the best option compared to others to enhance heat transfer in trailing edge of the gas turbine blade with the pressure drop in the cooling duct in the acceptable range.

**Keywords:** Gas turbine, Two-pass channel, Heat transfer, Computational Fluid Dynamic.
SAMMANFATTNING


Utmaningen är att utforma en inre kylkanal, i vilken kylmediet kommer in och lämnar bladet i rotsnittet som är tillräckligt bra för att hålla metalltemperaturen på normala värden utan att öka profilförluster och med acceptabla kylluftflöden och tryckfall.


Resultaten visar att för en två-pass rektangulär kanal (både släta och ribbade), minskar tryckfallet när inloppskanalens geometri reducerades. För en slät kanal påverkar inte den minskade bredden på inloppskanalens värmeöverföring i inlopps- och utloppskanalerna. Vid ribbade kanaler
minskar värmeöverföring vid toppen och på toppväggen med minskad bredd på inloppskanalen. Av de olika turbulensmodeller som används för attvalidera numeriska resultat mot experimentella för fallet med trapetsformad kanal visade sig låg-Re k-epsilon modellen den mest lämpliga. Genom att använda den turbulensmodell som är närmast experimentella data visar det att geometrin med förskjutna ribbor på bakkantsväggen har maximal termiska prestanda. Resultaten från det sammansatta värmeöverföringsproblemet framhåller användning av ånga som kylmedium om den finns tillgänglig eftersom den kräver mindre massflöde för att få samma värden på väggtemperaturerna jämfört med luft vid samma termiska tillstånd. Det kunde också visas att förskjutna turbulensribbor är det bästa alternativet jämfört med andra för att öka värmeöverföringen i bakkanten av ett gasturbinblad med acceptabelt tryckfall i kylkanalen.

**Nyckelord:** Gasturbiner, Två-pass kanaler, Värmeöverför, Computational Fluid Dynamic.
PREFACE

This thesis is based on four publications which are listed below and appended at the end of the thesis.

Paper I:
Siddique W.; El-Gabry L.; Shevchuk I.V.; Hushmandi N.B.; Fransson T.H.; 2011

Paper II:
Siddique W.; Shevchuk I.V.; El-Gabry L.; Hushmandi N.B.; Fransson T.H.; 2011

Paper III:
Siddique W.; El-Gabry L.; Shevchuk I.V.; Fransson T.H.; 2011
“Validation and Analysis of Numerical Results for a Two-Pass Trapezoidal Channel with Different Cooling Configurations of Trailing Edge”, Accepted for publication in ASME Journal of Turbomachinery. Manuscript ID: Turbo-11-1168.

Paper IV:
Siddique W.; El-Gabry L.; Fransson T.H.; 2011

Contribution of the various authors is as follows:

Paper I and II: The research idea was put up by the first author who also worked on the numerical simulations and was the main author. Second and third authors reviewed the numerical results. Fourth author was the numerical mentor. Fifth author acted as the initiator and was the main academic supervisor.
**Paper III:** The first author was the main author and responsible for the research idea as well as worked on all numerical simulations. Second and third author reviewed the numerical results. Fourth author acted as main academic supervisor.

**Paper IV:** The first author was the main author and responsible for the research idea as well as worked on all numerical simulations. Second author reviewed the numerical results. Third author acted as main academic supervisor.
ACKNOWLEDGEMENT

The author is grateful for the financial support offered by Higher Education Commission of Pakistan and Division of Heat and Power Technology, KTH Sweden. This work is also supported by grants of computing time from Centre of Parallel Computers (PDC) at the Royal Institute of Technology (KTH), Stockholm which is distributed by the Swedish National Allocation Committee.

I would like to express gratitude to my academic supervisor Prof. Torsten Fransson at the Chair of Heat and Power Technology, the Royal Institute of Technology (KTH), Stockholm whose guidance and encouragement made this work possible and realizable.

I would also like to acknowledge Prof. Torsten Strand at the Royal Institute of Technology (KTH), Stockholm and Dr. Esa Utriainen at Siemens Industrial Turbomachinery Sweden for the valuable discussions on various aspects of the thesis work.

From problem identification till conclusions of the results, Dr. Igor V. Shevchuk at MBtech Group GmbH & Co. Germany and Dr. Lamyaa El-Gabry at American University in Cairo, Egypt has been a great support. Their guidance throughout the thesis work is highly acknowledged.

For guiding me with CFD and parallel computing, I am grateful to Dr. Narmin B. Hushmandi who has left the department after completion of her PhD study and Qaiser Waheed from department of Theoretical Physics, the Royal Institute of Technology (KTH), Stockholm. Also, for reviewing my thesis, I am thankful to Florian Fruth. Dr. Seksan Udomsri has been a great source of encouragement and guidance in all aspects of departmental issues.

I would like to thank all my colleagues at the Department of Energy Technology for providing a constructive and friendly working environment.

I am grateful to the motivation provided to me by my parents, brothers and sisters back in Pakistan who very patiently waited for my successful return.

Finally I am very thankful to my wife Seemab for her love, support and encouragement and refreshing company of my sons Shamikh and Aarib.
CONTENTS

ABSTRACT .................................................................................................................................... I
SAMMANFATTNING ............................................................................................................... III
PREFACE ...................................................................................................................................... V
ACKNOWLEDGEMENT ........................................................................................................ VII
LIST OF FIGURES .................................................................................................................... XI
LIST OF TABLES ..................................................................................................................... XV
NOMENCLATURE ................................................................................................................. XVI

1. INTRODUCTION ................................................................................................................ 1
   1.1 OBJECTIVE OF THIS THESIS .......................................................................................... 2
   1.2 THESIS OUTLINE ............................................................................................................ 3

2. COOLING METHODS FOR GAS TURBINE BLADES ....................................................... 5
   2.1 INTERNAL COOLING ........................................................................................................ 7
   2.2 DIFFERENT INTERNAL COOLING CONFIGURATIONS ................................................... 9
       2.2.1 Radial Flow .................................................................................................................. 9
       2.2.2 Multi-Pass Serpentine Cooling ................................................................................ 9
       2.2.3 Enhanced Internal Cooling .................................................................................... 10
   2.3 TRAILING EDGE COOLING ............................................................................................ 13
   2.4 STEAM AS COOLANT ...................................................................................................... 16
   2.5 CONJUGATE HEAT TRANSFER ...................................................................................... 18
   2.6 SUMMARY OF CHAPTER 2 ............................................................................................ 19

3. METHODOLOGY ............................................................................................................. 21
   3.1 RECTANGULAR CHANNEL ............................................................................................... 21
   3.2 TRAPEZOIDAL CHANNEL ................................................................................................. 22
   3.3 CONJUGATE HEAT TRANSFER ......................................................................................... 23
   3.4 OVERALL ........................................................................................................................ 24

4. NUMERICAL MODELS AND VALIDATIONS ................................................................... 25
   4.1 TWO-PASS RECTANGULAR CHANNEL ........................................................................ 25
       4.1.1 Boundary Conditions .................................................................................................. 25
       4.1.2 Two-Pass Smooth Rectangular Channel .................................................................. 26
       4.1.3 Two-Pass Ribbed Rectangular Channel .................................................................. 31
   4.2 TWO-PASS TRAPEZOIDAL CHANNEL .......................................................................... 35
       4.2.1 Description of the Physical Model .......................................................................... 36
       4.2.2 Boundary Conditions ............................................................................................... 37
       4.2.3 Grid Independence .................................................................................................... 38
       4.2.4 Validation .................................................................................................................. 39
   4.3 CONJUGATE HEAT TRANSFER ANALYSIS ................................................................... 42
       4.3.1 Description of the Physical Model .......................................................................... 43
4.3.2 Boundary Conditions ................................................................. 47
4.3.3 Design Approach ........................................................................ 48
4.3.4 Grid Independence ................................................................. 48

5. RESULTS FROM RECTANGULAR CHANNEL SIMULATIONS ............... 51

5.1 TWO-PASS SMOOTH RECTANGULAR CHANNEL ................................ 51
  5.1.1 Pressure Drop and Heat Transfer ................................................. 51
  5.1.2 Overall Comparison ................................................................. 55

5.2 TWO-PASS RIBBED RECTANGULAR CHANNEL ................................ 56
  5.2.1 Pressure Drop and Heat Transfer ................................................. 56
  5.2.2 Overall Comparison ................................................................. 60

6. RESULTS FROM TRAPEZOIDAL CHANNEL SIMULATIONS .................. 63

7. RESULTS FROM CONJUGATE HEAT TRANSFER SIMULATIONS ........... 71

  7.1 HEAT TRANSFER ................................................................. 72
  7.2 PRESSURE DROP ................................................................. 73
  7.3 VARIATION IN DIVIDER WALL ................................................... 74
  7.4 IMPINGEMENT AT CORNER ..................................................... 78
  7.5 THERMAL PERFORMANCE ..................................................... 83

8. CONCLUSIONS ............................................................................. 85

9. FUTURE WORK ............................................................................ 87

10. REFERENCES ............................................................................... 89

APPENDIX A GOVERNING EQUATIONS ................................................. 97

APPENDIX B CFD AND TURBULENCE MODELLING ............................... 99

  B.1 TURBULENCE ................................................................. 99
  B.2 REYNOLDS-AVERAGED NAVIER-STOKES (RANS) ......................... 99
  B.3 TURBULENCE MODELLING .................................................. 100
  B.3.1 THE BOUSSINESQ HYPOTHESIS ......................................... 100
  B.3.2 TWO EQUATION TURBULENCE MODELS ............................... 101
   B.3.2.1 the Low-Re k-ε Model ...................................................... 101
   B.3.2.2 the RNG k-ε Model ......................................................... 102
   B.3.2.3 the Realizable k-ε Model .................................................. 103
   B.3.2.4 the Low-Re k-ω model ..................................................... 104
   B.3.2.5 Shear-Stress Transport (SST) k-ω model ............................. 105
  B.4 WALL TREATMENT .............................................................. 106
  B.4.1 WALL CALCULATION METHODS ........................................ 108
  B.4.2 LIMITATIONS OF WALL FUNCTION APPROACH .................. 109
  B.4.3 ENHANCED WALL TREATMENT ......................................... 109
LIST OF FIGURES

Figure 2-1: Improvements of material and cooling technology [Rubensdörffer; 2006] ______________________ 5
Figure 2-2: Schematic of air-cooled turbine, with cross-section of cooled airfoil section at top [Sarvanamutto et al.; 2005] _______________________________ 6
Figure 2-3: Typical airfoil cooling schemes for high-pressure turbine blade [Chuna; 2006] __________________ 7
Figure 2-4: Example of a turbine blade internal cooling system [Bell et al.; 2009] __________________________ 8
Figure 2-5: Example of a turbine blade internal cooling system [Iacovides; 2006] __________________________ 8
Figure 2-6: Schematic of multi-pass serpentine passages for internal cooling of turbine blade [Obata; 1989] ___ 10
Figure 2-7: A ribbed channel [Han; 2000] _____________________________________________________________ 10
Figure 2-8: Flow separation and reattachment around ribs [Sundberg; 2006] ____________________________ 11
Figure 2-9: Flow structure for different rib pitch-to-rib height ratios [Cunha; 2006] ________________________ 13
Figure 3-1: Layout of the steps involved in rectangular channel simulation. _______________________________ 22
Figure 3-2: Layout of the steps involved in trapezoidal channel simulation. _______________________________ 23
Figure 3-3: Layout of the steps involved in conjugate heat transfer simulation. ___________________________ 24
Figure 4-1: Schematic view of the smooth two-pass channel with \( W_{\text{in}}/H = 1:3 \) at inlet pass and \( W_{\text{out}}/H = 1:1 \) at outlet pass. ___________________________________________ 26
Figure 4-2: Three grids at the bend section of the channel with \( W_{\text{in}}/W_{\text{in}} = 1.5 \). ______________________ 27
Figure 4-3: Comparison of the average Nusselt numbers in different regions for the three cases of the smooth two-pass channel with \( W_{\text{in}}/W_{\text{in}} = 1.5 \). ________________________________ 28
Figure 4-4: Local Nusselt number distribution by experimental (left) and numerical (right) results. ________ 29
Figure 4-5: Effect of \( W_{\text{in}} \) on average heat transfer in the inlet pass (faces 1a and 2), bend (face 3 and 4) and outlet pass (faces 6a and 5) as predicted in current simulation and by Shevchuk et al [2011] _________________ 31
Figure 4-6: Schematic view of the ribbed two-pass channel with \( W_{\text{in}}/H = 1:3 \) in the inlet pass and \( W_{\text{out}}/H = 1:1 \) at outlet pass and a symmetry boundary condition on the upper surface. ______________________________________ 32
Figure 4-7: A section of the three grids used for grid independence study. ________________________________ 32
Figure 4-8: Comparison of the average Nusselt numbers normalized with Dittus-Boelter correlation in different regions for the three cases. _______________________________________________ 33
Figure 4-9: Local Nusselt number distribution from experimental (left) and numerical (right) results. ________ 34
Figure 4-10: Schematic view of the three designs for augmented heat transfer at trailing edge. ______________ 37
Figure 4-11: Three grids used for grid independence study. _____________________________________________ 38
Figure 4-12: Segment numbers at the bottom wall of the channel. ______________________________________ 39
Figure 4-13: Comparison of numerical results from \( k-\epsilon \) models and experimental data for the smooth channel. _______________________________ ________________ 39
Figure 4-14: Comparison of numerical results from \( k-\omega \) models and experimental data for the smooth channel. ___________________________________________________________ 40
Figure 4-15: Comparison of numerical results from the low-Re \( k-\epsilon \) model and SST \( k-\omega \) model with experimental data for the ribbed channel. __________________________________________________________ 41
Figure 4-16: Schematic view of the two-pass smooth channel (Case A). Inset is the front view of the same channel. __________________________________________________________ 44
Figure 4-17: Schematic view of the two-pass ribbed channel (Case B). Inset is the top view of a section of the same channel. ___________________________________________________________ 45
Figure 4-18: Schematic view of the two-pass channel with ribs at the trailing edge only (Case C). Inset is the close up of the same channel. ________________________________________________ 45
Figure 4-19: Schematic view of the two-pass channel with ribs at the trailing edge as well as at the bottom wall with inline arrangement (Case D). Inset is the close up of the same channel. ____________ 46
Figure 4-20: Schematic view of the two-pass channel with ribs at the trailing edge as well as at the bottom wall with staggered arrangement (Case E). Inset is the close up of the same channel.  

Figure 4-21: Front view of the two-pass smooth channel with a small fin at the trailing edge (Case F, G and H).  

Figure 4-22: Front view of the two-pass ribbed channel with fin at the trailing edge (Case I).  

Figure 4-23: Grid independent study for two-pass smooth channel.  

Figure 4-24: Grid independent study for two-pass ribbed channel.  

Figure 5-1: Effect of W_{el} normalized with the height of the channel H (on the left) while normalized with inlet pass width W_{in} (on the right) on the relative pressure drop for two-pass smooth channel.  

Figure 5-2: Heat transfer distribution for smooth two-pass channel with aspect ratio of W_{in}/H = 1:3 and W_{out}/H =1:1 for different W_{el}/W_{in}.  

Figure 5-3: Effect of W_{el} on the average heat transfer in the inlet pass (faces 1a and 2), bend (faces 3 and 4) and outlet pass (faces 6a and 5), for two channels of different inlet aspect ratios.  

Figure 5-4: Velocity magnitude distribution for two channels with W_{el}/W_{in} = 1.5.  

Figure 5-5: Thermal performance at bend (faces 3 and 4) of the smooth two-pass channel with aspect ratio of W_{in}/H = 1:3 and W_{out}/H =1:1 for different W_{el}/H.  

Figure 5-6: Effect of W_{el} on the relative pressure drop for two-pass ribbed channel.  

Figure 5-7: Heat transfer distribution over the bottom, tip and external side walls of the ribbed two-pass channel with varying aspect ratio of W_{in}/H = 1:3 (inlet) and W_{out}/H =1:1 (outlet), W_{el}/H= 0.88.  

Figure 5-8: Nusselt number distribution at different zones of two-pass ribbed channel.  

Figure 5-9: Effect of W_{el} on average heat transfer on the bend bottom (face 3) and tip wall (face 4 in the ribbed channel, different varying W_{el}/H.  

Figure 5-10: Effect of the W_{el} on average heat transfer on different faces in the ribbed channel, varying W_{in}/H = 1:3 to W_{out}/H = 1:1.  

Figure 5-11: Contours of Velocity magnitude near the bottom wall and at symmetry plane of two different divider-to-tip wall distances for channel with W_{in}/H = 1:3.  

Figure 5-12: Thermal performance at different divider-to-tip wall distances for channel with W_{in}/H = 1:3.  

Figure 6-1: Contours of Nu/Nu_0 for the two-pass smooth trapezoidal channel.  

Figure 6-2: Contours of Nu/Nu_0 for the two-pass ribbed trapezoidal channel.  

Figure 6-3: Contours of Nu/Nu_0 for the two-pass ribbed trapezoidal channel with ribs on the trailing edge only (Case A).  

Figure 6-4: Contours of Nu/Nu_0 for the two-pass trapezoidal channel with ribs on the bottom wall as well as on the trailing edge with inline arrangement (Case B).  

Figure 6-5: Contours of Nu/Nu_0 for the two-pass trapezoidal channel with ribs on the bottom wall as well as on the trailing edge with staggered arrangement (Case C).  

Figure 6-6: Area averaged Nu/Nu_0 for outlet pass of different configurations of the trapezoidal channel.  

Figure 6-7: Pressure drop across different configurations of the trapezoidal channel.  

Figure 6-8: Thermal performance for outlet pass of different configurations of the trapezoidal channel.  

Figure 7-1: Normalized temperature profile at trailing edge corner for two-pass smooth channel at different Reynolds number for air and steam.  

Figure 7-2: Temperature profiles at trailing edge corner for all cases.  

Figure 7-3: Pressure drop across different channels.  

Figure 7-4: Two-pass smooth channel with tapered and tilted divider wall.  

Figure 7-5: Velocity magnitudes at the symmetry plane of two-pass smooth channel with tapered and tilted divider wall.  

Figure 7-6: Temperature profile at trailing edge corner for a smooth channel and its two variations of divider wall.  

Figure 7-7: Velocity magnitudes at the symmetry plane of two-pass channel with staggered arrangement of ribs and tilted divider wall.
Figure 7-8: Temperature profile at trailing edge corner for a staggered ribbed channel and its variation of divider wall. 77

Figure 7-9: Normalized temperature contours at different walls of the channel with tilted divider wall and staggered arrangement of ribs. 78

Figure 7-10: Top view of three different cases of impingement cooling of smooth channel. 79

Figure 7-11: Contours of velocity magnitude at symmetry plane for three different cases of impingement cooling of smooth channel. 79

Figure 7-12: Normalized temperature profile at trailing edge corner for three cases of impingement cooling of smooth channel. 80

Figure 7-13: Contours of normalized temperature at different walls for three cases of impingement cooling of smooth channel. 81

Figure 7-14: Velocity magnitude at the symmetry plane of a channel having L-shaped divider wall with a tapered opening and staggered ribs in the outlet pass. 82

Figure 7-15: Normalized temperature profile at trailing edge corner for three different types of channels with staggered arrangement of ribs. 82

Figure 7-16: Contours of normalized temperature at different walls of the channel with staggered arrangement of ribs and impingement in the corner. 83

Figure B-1: Subdivisions of the Near-Wall Region [ANSYS FLUENT Theory guide; 2009] 107

Figure B-2: Near Wall treatments in ANSYS FLUENT [ANSYS FLUENT Theory guide; 2009] 108
LIST OF TABLES

Table 4-1: Comparison of average Nusselt numbers from experiments and numerical simulation for \( \frac{W_e}{W_m} = 1.5 \).

Table 4-2: The three configurations for augmented heat transfer at trailing edge.

Table 4-3: Pressure losses in the smooth and ribbed trapezoidal channels.

Table 4-4: Different two-pass channel configurations studied.

Table 4-5: Wall boundary conditions applied to all cases.

Table 6-1: Area averaged \( \frac{Nu}{Nu_0} \) and thermal performance for trailing edge of the trapezoidal channels.

Table 6-2: Thermal performance for the bottom wall of a smooth and a ribbed bottom wall channel.

Table 7-1: Normalized friction factor for three cases of impingement cooling of smooth channel.

Table 7-2: Pressure drop and thermal performance of different two-pass trapezoidal channels.
## NOMENCLATURE

### Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Area</td>
<td>[m²]</td>
</tr>
<tr>
<td>AR</td>
<td>Aspect Ratio</td>
<td>[-]</td>
</tr>
<tr>
<td>C</td>
<td>Specific heat</td>
<td>[J kg⁻¹ K⁻¹]</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
<td>[-]</td>
</tr>
<tr>
<td>CPU</td>
<td>Central Processing Unit</td>
<td>[-]</td>
</tr>
<tr>
<td>D</td>
<td>Diameter</td>
<td>[mm]</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
<td>[-]</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
<td>[-]</td>
</tr>
<tr>
<td>e</td>
<td>Rib height/width</td>
<td>[mm]</td>
</tr>
<tr>
<td>Exp.</td>
<td>Experimental</td>
<td>[-]</td>
</tr>
<tr>
<td>f</td>
<td>Friction factor</td>
<td>[-]</td>
</tr>
<tr>
<td>h</td>
<td>Heat transfer coefficient</td>
<td>[W m⁻² K⁻¹]</td>
</tr>
<tr>
<td>H</td>
<td>Height of channel</td>
<td>[mm]</td>
</tr>
<tr>
<td>I</td>
<td>Turbulence intensity,</td>
<td>[-]</td>
</tr>
<tr>
<td></td>
<td>Distance of divider wall from trailing edge</td>
<td>[mm]</td>
</tr>
<tr>
<td>k</td>
<td>Thermal conductivity of the fluid,</td>
<td>[W m⁻¹ K⁻¹]</td>
</tr>
<tr>
<td></td>
<td>Turbulence kinetic energy</td>
<td>[m² s⁻²]</td>
</tr>
<tr>
<td>L</td>
<td>Stream wise distance between locations of the pressure stations in the channels,</td>
<td>[mm]</td>
</tr>
<tr>
<td></td>
<td>Length of the channel</td>
<td>[mm]</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
<td>[-]</td>
</tr>
<tr>
<td>m</td>
<td>Mass flow rate</td>
<td>[kg s⁻¹]</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>M</td>
<td>Million</td>
<td>$10^6$</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
<td>$(h D_h k^{-1})$</td>
</tr>
<tr>
<td>P</td>
<td>Pitch between ribs</td>
<td>[m]</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
<td>[N m$^{-2}$]</td>
</tr>
<tr>
<td>Pe</td>
<td>Perimeter</td>
<td>[mm]</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
<td>$(C_p \mu k^{-1})$</td>
</tr>
<tr>
<td>r</td>
<td>Rib height</td>
<td>[mm]</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stokes</td>
<td>[-]</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>$(\rho U_{in} D_h \mu^{-1})$</td>
</tr>
<tr>
<td>RNG</td>
<td>Re-Normalization Group</td>
<td>[-]</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
<td>[-]</td>
</tr>
<tr>
<td>t</td>
<td>Thickness</td>
<td>[mm]</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>TBC</td>
<td>Thermal Barrier Coating</td>
<td>[-]</td>
</tr>
<tr>
<td>TIT</td>
<td>Turbine Inlet Temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>U</td>
<td>Velocity</td>
<td>[m s$^{-1}$]</td>
</tr>
<tr>
<td>W</td>
<td>Width</td>
<td>[mm]</td>
</tr>
</tbody>
</table>

**Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon$</td>
<td>Turbulence dissipation rate</td>
<td>[m$^2$ s$^{-3}$]</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>Difference</td>
<td>[-]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg m$^{-3}$]</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Thermal performance</td>
<td>[-]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Viscosity</td>
<td>[Ns m$^{-2}$]</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Specific dissipation rate</td>
<td>[s$^{-1}$]</td>
</tr>
</tbody>
</table>
### Subscripts

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Avg.</td>
<td>Average</td>
<td>[-]</td>
</tr>
<tr>
<td>c</td>
<td>Cross-section</td>
<td>[-]</td>
</tr>
<tr>
<td>el</td>
<td>Divider-to-tip wall</td>
<td>[-]</td>
</tr>
<tr>
<td>f</td>
<td>Fluid</td>
<td>[-]</td>
</tr>
<tr>
<td>h</td>
<td>Hydraulic</td>
<td>[-]</td>
</tr>
<tr>
<td>in</td>
<td>Inlet</td>
<td>[-]</td>
</tr>
<tr>
<td>I</td>
<td>Kays et al.</td>
<td>[-]</td>
</tr>
<tr>
<td>out</td>
<td>Outlet</td>
<td>[-]</td>
</tr>
<tr>
<td>o</td>
<td>Dittus-Boelter equation</td>
<td>[-]</td>
</tr>
<tr>
<td>pe</td>
<td>Petukhov et al.</td>
<td>[-]</td>
</tr>
<tr>
<td>s</td>
<td>Solid</td>
<td>[-]</td>
</tr>
<tr>
<td>st</td>
<td>straight channel</td>
<td>[-]</td>
</tr>
<tr>
<td>targ</td>
<td>Target</td>
<td>[-]</td>
</tr>
<tr>
<td>T</td>
<td>Total</td>
<td>[-]</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing Edge</td>
<td>[-]</td>
</tr>
<tr>
<td>web</td>
<td>Divider wall</td>
<td>[-]</td>
</tr>
</tbody>
</table>
1. INTRODUCTION

The gas turbine industry is always seeking to increase the thermal efficiency of the gas turbine. One of the ways is by increasing the turbine inlet temperature. Increasing the operating temperatures however leads to some major problems. The melting temperature of most materials is far less than the design operating temperatures of the gas turbine. In addition to that the rapid spatial variations in temperature within the blade can create thermal stresses which can be in dangerous limits. In order to tackle the problems related to thermal stresses, oxidation and creep which limit the lifetime of turbines, cooling of the blades is required.

To withstand the increased inlet temperature, different methods are used. These include use of ceramic material, use of thermal barrier coatings and cooling of the blade material. Because of their superior high-temperature strength and durability, ceramics can be used as structural materials for hot section components (blades, nozzles, combustor liners etc.). Efforts have been made on the development of silicon nitride and silicon carbide materials for small turbine rotors. Adequate reliability and life are difficult to achieve, but demonstrator engines have been run for short periods. Ceramic rotor blades have been investigated for the use in stationary gas turbines for power values up to about 5MW, and field tests were carried out in the late 1990s [Sarvanamutto; 2005]. About 1000 hours of endurance running were achieved, before the blade was destroyed by impact of a small object which broke loose within the combustor. It appears that ceramic blades are subject to brittle failure.

Thermal Barrier Coatings (TBC) insulates the components of the gas turbine which are operated at elevated temperature. Typical examples are turbine blades, combustor cans and nozzle guide vanes. An adherent layer of a low thermal conductivity material is attached to the surface of an internally cooled gas turbine blade, thus inducing a temperature drop across the thickness of the layer. It not only results in temperature reduction in the metal temperature of the component to which it is applied but also reduces the (thermal activated) oxidation rate of the bond coat applied to metal component and hence delays failure by oxidation.

Another method is to cool the turbine blades, where heat is dissipated by the coolant. The temperature of a blade can be reduced by increasing the surface area and also by increasing the coolant flow rate. The area is increased by adding fins or ribs to the surface. In case of gas turbine blades, conventionally, relatively cool air is bled from the compression system and is used as a coolant to dissipate heat from the blade, which prevents the blade from melting. There are many types of cooling methods, which includes internal air cooling, external air cooling, liquid spray cooling etc.

Although the use of thermal barrier coating and ceramic material helps in achieving higher inlet gas temperatures but still cooling of the blade is the most adopted method due to less cost.
involves. Also, it can be used in combination with thermal barrier coating. Therefore research has been carried out extensively over the years to improve the cooling methods. This includes numerical as well as experimental work related to different heat transfer enhancement methods in the blade. The availability of steam in combine cycle power plants provides an opportunity to use it as coolant instead of air. This requires the complete internal cooling design of different sections of blade including the trailing edge.

1.1 Objective of this thesis

The conventional method to cool the blade is to use air from the compressor. The fraction of air used as coolant bypass the combustor. Thus, the diverted coolant air does not receive energy directly from the combustors and does not completely expand through the turbine and therefore ultimately results in an adverse effect on the overall efficiency of the turbine. Therefore, alternative coolants should be used to avoid usage of air from the compressor as coolant.

Steam has a good potential as coolant compared to air. It is available in case of combine cycle power plant so can be used as coolant instead. The steam is required as working fluid for the steam cycle as well, therefore the steam used as coolant is not allowed to leave the blade. Hence, an internal cooling mechanism has to be designed. The trailing edge of the gas turbine blade requires special attention because of its small thickness which is forced by aerodynamics. This makes the internal cooling of the trailing edge as a challenging task.

The main objectives of the current work are as follows:

1. To understand the heat transfer and flow physics in internally cooled two-pass smooth and ribbed channel and to predict the effect of reducing the width of the inlet pass on heat transfer in bend and outlet pass of the channel as well as the pressure drop across the channel. The purpose of reduction in width is to increase the number of channels in the blade profile in chord-wise direction which will have positive effect on heat transfer due to increased number of turns.

2. To design an internal cooling configuration for a trailing edge, modelled as a trapezoidal channel, in order to enhance heat transfer with minimal pressure drop. This requires the analysis of different rib configurations in the trapezoidal channel.

3. To design a cooling configuration for the trailing edge using internal convection only, which has not been done before. The aim is to use conjugate heat transfer technique to find suitable configuration which is effective in limiting the maximum wall temperatures below a certain value for engine-similar thermal conditions. These include the heat transfer coefficient and the gas temperature. This accompanies the analysis of steam as a better coolant than air under given conditions.
1.2 Thesis outline

This thesis presents results based on the published worked in paper I to IV appended. These results make basis for internal cooling designs for the trailing edge of the gas turbine blade. Chapter 2 presents the overview about the internal cooling of the gas turbine blade and enhancement of heat transfer by use of rib turbulators. It also includes the literature survey on trailing edge cooling, conjugate heat transfer analysis and usage of steam as coolant. This helps in building the background for achieving the goals set as objectives. Chapter 3 presents the methodology adopted in the current thesis work. It is divided into three parts covering all aspects of the objectives defined. Chapter 4 summarizes the numerical models used and the validation of numerical results for heat transfer and pressure drop in rectangular and trapezoidal two-pass channels. This chapter also includes the numerical model used to study the conjugate heat transfer problem. Chapter 5 summarizes the results for reduction of inlet pass width in a rectangular two-pass channel, based on paper I and II. Chapter 6 presents selected results of heat transfer and pressure drop in a two-pass trapezoidal channel, based on paper III. Chapter 7 summarizes the results of conjugate heat transfer problem and steam cooling, based on paper IV. Chapter 8 gives the conclusions of the thesis work. Suggestions for future work are given in Chapter 9 while reference can be found in chapter 10.
2. COOLING METHODS FOR GAS TURBINE BLADES

The thermal efficiency as well as power output of a gas turbine increases with the gas temperature which leaves the combustor. Today gas turbines are being operated at a Turbine Inlet Temperature (TIT) as high as 1850 K. Since modern alloys can withstand up to about 1350 K, an efficient cooling of the turbine parts is required to protect the blades and vanes from thermal failure. Over the years, designers have developed methods to increase heat transfer in the blade. Figure 2-1 shows the historical progress of turbine inlet temperatures with time for a Rolls-Royce family of gas turbines. Since 1960’s, with the introduction of cooling, the rate of increase of turbine inlet temperature in actual engines is faster than that permitted by advances in material technology. The difference in slope of cooled and uncooled blades indicates the benefits of blade cooling. The “demonstrator capability” line in the figure indicates the potential of blades to withstand more increased turbine inlet temperatures, through recent and future research.

![Figure 2-1: Improvements of material and cooling technology [Rubensdörffer; 2006]](image)

Gas turbine blades are cooled with the combination of a number of traditional cooling concepts. Figure 2-2 shows a schematic of air-cooled turbine. Air, bled from the compressor, is introduced into the turbine blades through their roots. On the way, it cools the rim of turbine disc. The coolant is extracted from the compressor, so less mass flow is available for combustion. Also, to accommodate coolant in the trailing edge, the thickness of the trailing edge has to be increased. This increase leads to creation of a thick wake behind the trailing edge which has negative effects on the aerodynamic. Therefore, it is necessary to optimize the cooling technique. The coolant can be used in different ways, such as it can be impinged on the inner surface of leading
edge called impingement cooling, convective internal cooling of blade surface and to cover the blade with a film of cool air, called film cooling. All the three can be seen in the Figure 2-2 as 1, 2 and 3 respectively.

![Schematic of air-cooled turbine, with cross-section of cooled airfoil section at top](image)

**Figure 2-2:** Schematic of air-cooled turbine, with cross-section of cooled airfoil section at top

[Sarvanamutto et al; 2005]

Figure 2-3 shows the more modern cooling technique with three internal cooling zones in a turbine blade. As shown in the Figure, the blade is hollow, such that cooling air can pass through it internally, in channels. The main internal channel consists of serpentine cooling passages which are ribbed roughened. Leading edge is cooled by impinging coolant on its inner surface. The trailing edge is provided with fins which are pin shaped. These pins results in enhancement of heat transfer as do the ribs in main passage. The coolant enters the blade from its roots, passes through the main serpentine rib roughened passage and impinges on the inner surface of the blade leading edge. Then part of the coolant air leaves through the trailing edge and tip. A small amount of coolant air, after impinging on the walls of the airfoil, exits the blade and provides a protective film on the vane’s external surface. This is known as film cooling. This film is a thin but cool insulating blanket along the external surface of the turbine blade is formed by this air. As a result, the blade is able to sustain higher turbine operating temperature and achieve longer lifetime [Han; 2000].
Overall the cooling techniques can be divided into two main classes, Internal Cooling and External Cooling (film cooling). When steam is used as a coolant, it is not allowed to mix with the main gas flow, thus eliminating any chances of using film cooling. This makes the internal cooling technique the only method to be used for heat transfer. Internal cooling and ribbed augmentation technique is discussed in following paragraphs.

2.1 Internal Cooling

Internal cooling, as depicted by its name, cools down the blade by the flow of a fluid inside a cavity, provided in a blade or vane of a turbine. The main aim is to decrease the temperature of the blade surface by increasing the heat transfer from the blade surface to the coolant. Air is drawn from the compressor and is fed through serpentine passages within the coolant passages of the airfoil. This removes the heat by convection. In most cases, the coolant is then ejected at the blade tip, through trailing edge cooling slots or through film cooling holes on the airfoil surface. An example of a turbine blade internal cooling system is shown in Figure 2-4.
In order to utilize minimum amount of mass flow for efficient cooling, the blades are provided with features that increase the heat transfer coefficient. The heat transfer coefficient is increased by enhancement of the flow turbulence and by breaking the flow boundary layer. The negative part is the increased pressure drop. An example of an internally cooled blade with such enhancement features is shown in Figure 2-5. On the left is blade where the mid span has cooling channels. The coolant flows in these channels and extracts heat from the wall and keep the wall temperatures in acceptable range. The trailing edge has pin-fins, which act as heat transfer enhancers. Coolant enters from the roots of the blade and leaves from the tip. On the right hand side, another blade is shown where ribs are provided in the cooling channel which enhances heat transfer.

**Figure 2-4:** Example of a turbine blade internal cooling system [Bell et al.; 2009]

**Figure 2-5:** Example of a turbine blade internal cooling system [Iacovides; 2006]
Predicting the heat transfer coefficients, coolant and blade temperatures in these channels is very difficult. The complexities involved in such flows are:

- Complex and narrow coolant passages
- Large turbulent mixing and separation associated with cooling jets and cavities
- Continuous change in the coolant’s temperature through its path which results in change in physical properties of the coolant

So, the analysis and design of internal cooling passages is highly empirical and based on experimental data. But, numerical techniques have developed a lot in the recent years. This has resulted in a decreased use of empirical relationships and increased the accuracy of prediction.

### 2.2 Different Internal Cooling Configurations

Different configurations of internal cooling that have been in use are described below.

#### 2.2.1 Radial Flow

Due to cost and manufacturing constraints, initial cooling systems were forced to be based on radial hole schemes where radial holes inside the blade profile were produced. Forced convection through the radial holes may not be sufficient for high thermal load applications. Even at low thermal loads, this simple cooling arrangement is likely to induce high temperature gradients between the airfoil surface and the cooling hole locations.

#### 2.2.2 Multi-Pass Serpentine Cooling

An improved cooling arrangement compared to the radial cooling hole arrangement is the multi-pass serpentine cooling configuration. Figure 2-6, shows one such arrangement. Here the coolant enters the airfoil through the blade root inlet. Than it passes through multiple circuits, cooling the mid-body of the airfoil before being ejected out of the airfoil through the root again, In addition to that, for more advance cooling design the coolant also leave the blade through the main body film holes or trailing edge slots.
2.2.3 Enhanced Internal Cooling

With increasing gas inlet temperature, a need was felt for more different ways to cool gas turbine blades. Therefore, keeping the basic concept the internal cooling intact, additional enhancement techniques were implemented. These includes introduction of ribs in the mid-span of the blade, allowing coolant to impinge at the leading edge (and in some cases at trailing edge) and placing pins at the trailing edge. There have been many studies which focus on the effect of these and many other enhancements on heat transfer and pressure drop. This thesis focuses on the rib turbulated enhancement of heat transfer, therefore only this is discussed in detail here.

Rib Turbulated Cooling

To enhance the heat transfer in advanced gas turbine blades, repeated rib turbulence promoters are cast on the two opposite walls (pressure and suction sides) of internal cooling passages. To match the external loads, which can be different at pressure and suction sides, the ribs sometimes can be provided only on one side. Figure 2-7, shows a section of ribs provided in the channel.
Due to presence of ribs, the flow separates at top of the rib and reattaches to the flow between the ribs. Figure 2-8 shows the flow separation and reattachment due to presence of ribs. The boundary layer is disturbed as well as the turbulence of the flow increases due to separation and reattachment. This mixes the fluid elements near the wall with the cooler ones in the middle of flow. The two phenomena results in enhancement of heat transfer.

![Figure 2-8: Flow separation and reattachment around ribs [Sundberg; 2006]](image)

Thermal energy is transferred from the external pressure and suction surfaces of the turbine blades to the inner zones through conduction and that heat is removed by internal cooling. The internal cooling passages are mostly modelled as short rectangular or square channels with different aspect ratios. Chandra et al. [1988], Han [1988] and Han et al. [1992] have showed that there are certain geometrical parameters that affect the heat transfer coefficient. Such parameters include the passage aspect ratio, blockage ratio, rib angle of attack, reciprocal ribs positioning, rib pitch to height ratio and the rib shape. There have been many fundamental studies to understand the heat transfer enhancement phenomena by the flow separation caused by ribs. Han [1984] investigated the effect of rib configuration (such as rib height, spacing, angle of attack) on the average heat transfer and friction in a straight, square channel with two opposite rib-roughened walls. Han and Park [1988] investigated these effects on straight rectangular channels of different aspect ratios. Han et al. [1989] reported the local heat transfer distributions in short rectangular channels of narrow aspect ratios (W/H 2/4 and ¼) with rib turbulators at opposite walls. If the ribs are inclined to the direction of core flow, the secondary flows are induced by the ribs in the core flow. This circulates the fluid from centre of the channel to the walls, thus increasing heat transfer along the walls. The effect of aspect ratios and rib angle of attack was investigated in this study. Park et al. [1991] compared the combine effect of rib angle of attack and flow Reynolds number on heat transfer performance in five rectangular channels with different aspect ratios. The channel aspect ratios were ¼, ½, 1, 2 and 4. It was concluded that 30° and 45° angled ribs with high aspect ratio (W/H=4:1) enhances more heat transfer. Rau et al. [1998] measured the local heat transfer performance in a ribbed roughened square channel with blockage ratio (rib height to hydraulic diameter of the channel) of 10 to 20%. They found that the strong secondary flows results in a three-dimensional flow field with high gradients in the local
heat transfer distributions on the smooth side walls. They concluded that the correlation used for predicting heat transfer in a ribbed channel with small blockage ratio is not applicable in case of high blockage ratio channel. Wang et al. [2001] performed experiments to study the local heat transfer and pressure drop characteristics of developing turbulent flows of air in three different types of ribbed ducts. These include the constant cross section square duct, the diverging square duct and the converging square duct. In comparison, the diverging duct showed the highest heat transfer characteristics while constant cross section duct was found better than the converging duct. Another study for heat transfer and friction losses in a square channel with 90° ribs on one, two, three and four walls was conducted by Chandra et al. [2003]. They performed the experiments with the Reynolds numbers varying from 10,000 till 80,000. It was found that with increase in ribbed walls the thermal performance decreases due to increase in pressure drop. Wright et al. [2004] studied the thermal performance of three different types of ribs (45° angled, V- and W-shaped) in a high aspect ratio (W/H= 4) channel with Reynolds number varied from 10,000 to 40,000. They found that W-shaped ribs performed better than the other two types of ribs. Gupta et al. [2008] experimentally investigated the heat transfer distribution for 90° continuous, 90° saw-tooth profiled and 60° V-broken rib configurations for Reynolds numbers ranging from 10,000 to 30,000. It was found, that for the same rib height to hydraulic diameter, 90° continuous and 90° saw tooth profiled configurations show no significant difference. But 60° V-broken ribs perform better than the others.

Kunstmann et al. [2009] performed experiments to investigate the thermal performance of W-shaped, 2W-shaped and 4W-shaped ribs in straight high aspect ratio rectangular channels. The aspect ratios were 2:1, 4:1 and 8:1. They found that pressure losses increases with increasing complexity of rib geometry. For the investigated rib geometries, W-shaped ribs performed best for channel with an aspect ratio of 2:1, 2W-shaped ribs performed best for channel with an aspect ratio of 4:1 and 4W-shaped ribs performed best for channel with an aspect ratio of 8:1. Annerfeldt et al. [2001] used thin foil heaters, combined with an infrared camera to compare the Nusselt number for different surfaces provided with different shapes of tabulators like triangular, V-shaped ribs and others. Results showed that the heat transfer increased by a factor of 1-1.3 compared to the flat plate.

The coolant generally flows along the main channel axis in case of a standard rib-roughened channel. But coupling internal forced convection with impingement cooling can maximize the effectiveness. Such studies have been done by Cho and Goldstein [1995] font, Huang et al. [1998] and Pamula et al. [2001].

The distance between the ribs, called rib pitch, has an effect on separation and reattachment of the flow, thus it affects the heat transfer enhancement in the channel. Han et al. [1978], has experimentally investigated that the maximum heat transfer is obtained for Pitch-to-rib height ratio of 10. Below this value the reattachment between the ribs was not visible. Figure 2-9,
shows the separation and reattachment of flow phenomenon for different Pitch-to-rib height ratios (P/e). For P/e less than 7, there exist a separation bubble on top of each rib. When P/e is equal to 7, the boundary layer between the separation zones and on the ribs does not develop. For P/e equal to 10, which is believed to be the optimal value as maximum heat transfer is achieved, the re-attachment occurs between the ribs while the separation bubble at top of the ribs extend past the back of the rib and combine with the separation region following the rib. Thus at P/e = 10, maximum break-up of near wall flow is achieved which results in increase in turbulence level and enhance the exchange of fluid in the near wall region with the core flow.

![Flow structure for different rib pitch-to-rib height ratios](Cunha: 2006)

The ribs provide resistance to the flow, so causes pressure drop. This can be termed as the disadvantage associated with ribbed ducts. But, in many cases, due to relatively small ribs this pressure drop is acceptable.

### 2.3 Trailing Edge Cooling

The cooling of a complete blade is a difficult task, but the most difficult region is the trailing edge. The treatment is difficult not only due to heat transfer problems but also due to the aerodynamic losses. The trailing edge needs cooling but at the same time structural rigidity. Trailing edge cooling designs are tradeoffs between aerodynamic efficiency and heat transfer effectiveness. The aerodynamic losses associated with the trailing edge enforce the requirement of a narrow and smooth trailing edge. Thinning of the trailing edge of the blade reduces the
intensity of edge wake. This results in reduction of energy losses for mixing. This low thickness of the trailing edge results in the limitation of the magnitude of the conduction to internal cooling passage as well as the size of the passage. The convective heat flux is very high in this region, which can result in the production of burn marks, cracks and buckling. On the one hand the cooling of the trailing edge is essential, which leads to a thicker leading edge; the thickness of the blade trailing edge has a negative influence on the aerodynamic losses, on the other hand. So therefore a compromise is forced between blade cooling and aerodynamic losses. The loss associated with the trailing edge thickness is the profile loss which is a result of mixing processes associated with the velocity deficit on the wake and the trailing edge separation. These mixing processes are enhanced by the thickness of the trailing edge [Brundage et al.; 2007].

The most common method of cooling this region of the turbine blade is the injection of a film of cooling air through slots located on the airfoil pressure side near the trailing edge. This provides the thermal protection to the trailing edge by providing a cooling buffer between the hot mainstream gas and the airfoil surface [Cakan; 2007]. Cunha et al. [2006] also discussed the importance of the trailing edge heat transfer. The difficulty in cooling this region is in the relatively small area of the airfoil. The thickness of the region cannot be increased due to aerodynamic constraints. The other problem is that the heat capacity of a small trailing edge mass is different from the other parts of airfoil, so its thermal response is quick. This can lead to the cyclic loading. Obata et al. [1989] have developed a design methodology which is applicable to closed return-flow gas turbine blades that have a number of incoming and outgoing cooling passages of uniform cross-section laid span wise from root to tip. This includes an analytical method for calculating the cooling performance of a closed return flow gas turbine blade which is internally cooled. It was compared with experimental results and was found applicable to superheated steam near saturation. The results showed that the metal temperature remain uniform over the blade chord except the trailing edge, where it drops.

Due to wide application of the channel flow heat transfer, there has been a good amount of dedicated work done in heat transfer enhancement in different types of cooling channels. Initially square and rectangular channels were investigated but with time research began in the field of triangular and trapezoidal channels as well. These represent the trailing edge region of the blade. According to Wright and Ghoardani [2008] the results for triangular and trapezoidal cooling channels are contradictory. The results for equilateral triangles showed a similarity towards the results of square and circular tubes. As far as triangular channels with small apex angles are concerned, they vary a lot from circular tubes but smooth tube correlations can be used for predicting trends in trapezoidal channels. They investigated furthermore the heat transfer coefficients for rectangular and trapezoidal channels where both can be with a smooth or rib roughened walls. They reported that the Dittus-Boelter correlation can be used for rectangular channels but it over predicts for trapezoidal channel. When ejecting air from the trailing edge, the heat transfer coefficient decreases. Placing V-shaped ribs, results in an increase in heat transfer. Taslim et al. [1995], have studied the effect of bleed holes on the heat transfer in
trapezoidal passages. They showed that the averaged heat transfer coefficients were adequately predicted with the Dittus-Boelter correlation. In absence of trailing edge ejection, significant span-wise variation was noted which was eliminated with introduction of trailing edge ejection by the lateral flow. Hwang and Lu [2001] also published a similar result. They also pointed out that with an increase in ejection rate the heat transfer coefficient increases on the narrow side of channel but it reduces on the wide side. Kiml et al. [2001] noted similar effects for rectangular channel. Moon et al. [2002] studied the local distributions of the heat transfer coefficient on the walls at the turn of a smooth two-pass channel with a trapezoidal cross section for various rates of airflow through the channel. The heat transfer was found higher at the turn and the outlet pass. Lee et al. [2007] used the naphthalene sublimation technique to study the heat (mass) transfer distribution in a two-pass trapezoidal channel with 180° turn. The results were obtained over a range of Reynolds number for the channel with smooth walls and with ribs on one wall and on two opposite walls. They found that for all cases, the average heat transfer was higher on the downstream of the turn compared to that on the upstream of the turn. Experimental and computational study of a trapezoidal channel which simulates the trailing edge was performed by Coletti et al. [2010]. The channel was divided into two sections. The flow from the inlet-section, which was smooth, enters the ribbed outlet-section through tilted slots provided on one side wall. The flow eventually exits through straight slots on the opposite side wall. It was found that the jet formed from the first tilted slot impinges on the edge of the rib on the bottom surface of the outlet-section, resulting in a high heat transfer region. Due to decrease in mass and momentum of jets formed by the next slots, the effectiveness of the impingement decreases. Vertical flow structures are formed due to rib-jet interaction which results in high heat transfer regions at the upper surface too.

The 3D fluid flow and heat transfer patterns in case of the trapezoidal cross-section two-pass channel differs from the square or rectangular cross-section two-pass channels. Cravero et al. [1998] analyzed the flow field and heat transfer in a three-pass trapezoidal channel and showed that the geometry of the channel has strong influence on flow field especially at the regions of flow separation and recirculation. Taslim et al. [1997] investigated trapezoidal cooling channels and showed that the trapezoidal channel has higher thermal performance compared to the square channel. It was concluded that the stronger interaction of the adjacent walls results in an increased heat transfer of the trapezoidal duct. Ekkad et al. [2000] and Murata et al. [2004] investigated heat transfer in straight and tapered (from hub-to-tip) two-pass ribbed channels. They found that at low Reynolds numbers the heat transfer augmentation in the inlet pass is comparable, but at high Reynolds numbers the acceleration effect in the tapered channel leads to higher heat transfer as compared to the straight channel. At the outlet pass the heat transfer was found comparable. Kiml et al. [2003] studied the rib-induced secondary flow structure inside a trapezoidal channel with rib height proportional to the channel cross-section (proportional ribs) and constant height ribs (non-proportional ribs) at four rib inclinations i.e., 90°, 75°, 60° and 45°. They concluded that the proportional ribs offer less pressure losses, but they deteriorate the strength of the secondary flow rotational momentum as a result of wider space for the air flow
between the rib and the opposite wall. Also, the strength of the secondary flow rotational momentum increases with a change of rib inclination from 90° to 45°.

### 2.4 Steam as Coolant

The traditional approach for cooling gas turbine blades and nozzles is to extract air from a source at a sufficiently high pressure, e.g. by extracting air from the intermediate and last stages of the compressor. The cooling air flow circuits bypass the combustors, where heat is supplied to the thermodynamic cycle. Thus, the diverted coolant air does not receive energy directly from the combustors and does not completely expand through the turbine. This arrangement leads to losses of the turbine output and degrades the overall performance efficiency. Kirillov et al. [1986] has calculated that the useful work of a plant will reduce by 1.3-1.5% if 1% of the air is bled from the compressor for cooling. It can increase up to 18-20% of loss in useful work, if the bleed air is increased to 12-14%. So alternative coolants should be investigated in order to get rid of the disadvantages of air as a coolant.

There are some studies on alternative coolants such as water and steam [Singh et al.; 1995] and [Louis et al.; 1983]. In addition to academic research, commercial power plant which use steam as coolant do exists at present time in the world. Mitsubishi has launched 17 combined cycle power plants since 1997, all of which utilize the steam from the steam cycle to cool stationary components of gas turbine. Despite of regular wear and tear, the hot gas path parts have been found in good condition [Power Engineering; 2009]. General Electric (GE) is producing turbines which use steam as coolant. GE’s H System combined cycle power plants incorporates steam cooling in addition to other advancements like advanced single-crystal materials and thermal barrier coatings. This system is capable of achieving 60% efficiency [GE Energy; 2009]. In a combined cycle, steam is available at several pressure and temperature levels. Coolant air in a gas turbine can be replaced by steam as it is then possible to increase the firing temperatures in the combustion chamber. The use of superheated steam as a coolant provides some performance advantages, since steam can absorb more heat than air. Steam cooling of turbine blades is expected to allow the turbine inlet temperature to be increased beyond the temperature at which the turbine material can be used without cooling or with air-cooling thus increasing the cycle efficiency and power output [Sarvanamutto; 2005]. According to Arsenyev et al. [1994] it is possible to use the heat of cooling (thermal energy extracted by coolant) in the combined cycle power plant which is another advantage of using steam as coolant.

Li et al. [2001] provided information about the enhancement of the heat transfer by using mist/steam flow. Their arguments were that the latent heat of evaporation serves as a heat sink to absorb large amounts of heat. The heat sink effect reduces the bulk temperature and increases the temperature gradient near the wall, which further increases the heat conduction at the wall. The direct contact of a small amount of liquid droplets on the wall further increases heat transfer via direct wall-to-liquid heat conduction and results in an accelerated evaporation. The propulsive
momentum induced by wall-to-liquid droplet vaporization accelerates the transport of energy from the wall to the core flow. The flow mixing is increased by steam–particle interactions through particle dynamics. Among these effects, it has been found that the direct droplet deposit and evaporation play a dominant role.

Polezhaev [1997] has discussed the cooling for blades of high temperature gas turbines. According to him, for efficient convective heat transfer, the mass flow rate of the coolant should be high if air is used as coolant. In his work he proposed to use steam as coolant, which makes it possible to greatly reduce the coolant flow rate at constant blade temperature.

Cunha [1994] claims to invent a cooling scheme for blades in which steam and air cooling are integrated in a combined cycle system where the primary cooling is provided by steam while at off-design operating conditions for example, during start-up or an abrupt failure in the supply of steam, air is supplied as coolant. In accordance with that invention, an existing air-cooled gas turbine was modified to change over from operational air cooling to steam cooling.

MacDonald [2003] has reported that General Electric (GE) engineers designed a closed-circuit cooling system that uses steam instead of the traditional airfoil air cooling. Steam from the intermediate-pressure (IP), the superheater and the high-pressure (HP) steam turbine exhaust is used to cool the first- and second-stage nozzles and blades. Third-stage nozzles and buckets are air-cooled while the turbine’s fourth-stage is not cooled. After the thermal-energy-enhanced steam exits the second-stage nozzles and blades, it is re-circulated to the turbine’s reheater through the system’s cold-heat line, with the turbine effectively serving as a bottoming-cycle reheater.

Traditional combined cycle power plants use the energy of hot exhaust gases from the gas turbine to produce steam. This steam can then be used in another steam turbine cycle to extract energy. Wicks et al. [2002] has calculated the effect of gas turbine blades cooled by steam, which is later on used in the steam turbine cycle. The conclusion was that, the efficiency was marginally improved from 53.4 to 54.5%, but the power per pound per hour of compressor flow was increased by 28%.

Smith [2004] reported a temperature drop of 155°C over the first-stage air-cooled nozzle for a large industrial gas turbine. If the first-stage nozzle could be cooled with a closed-loop steam coolant without film cooling, the temperature drop would be less than 44°C, and hence, the work extraction from the turbine would increase. It was also concluded, that for achieving the same blade metal relative temperature, the required flow rate of air is more than that of steam.
Steam as a coolant is preferred in a closed loop. The reasons include:

- Avoiding wastage of steam to the hot gas flow and therefore reduce the cost.
- Mixing the cooled steam flow with the hot gas flow will result in a decrease in enthalpy of the hot gas (quenching effect) and hence engine performance.
- Mixing will lead to a disturbed aerodynamics of the flow.
- Taking advantage of the superheating condition of the steam flow at the blade exit and to extract work from it in the steam turbine.

### 2.5 Conjugate Heat Transfer

In case of gas turbine blades, the temperature gradient within the solid body and temperatures at the surface of the blade is important to obtain in order to design the cooling configuration which is useful for minimizing the maximum temperatures as well as the gradients. One method to obtain it is to perform a decoupled analysis of the blade external flow, the blade internal flow and the analysis of the heat conduction in the blade itself. This can be done by assuming a constant wall temperature and calculating heat transfer coefficients. These heat transfer coefficients can then be used to calculate the wall temperatures of the solid. The new wall temperatures can then be used to correct the heat transfer coefficients value. Several iterations may be required to get a converged solution. Alternatively, a conjugate heat transfer approach can be used, which refers to the interaction between the conduction inside a solid surface and the flow of fluid along it. A simple knowledge of the flow conditions (pressure, temperature, flow rate), outside and inside the blade can be used to obtain the temperature gradients and temperature values of the blade itself. It does not require any knowledge of the thermal boundary conditions on the blade wall.

The Conjugate heat transfer approach has been used by many researchers to predict the thermal conditions of the blade. Bohn et al. [1995a, 1995b, 1995c, 1999] have used the conjugate method in the area of cooled turbine blades using multi-block grids. These studies showed the validity of the conjugate approach in predicting the thermal conditions in the blade. Riby et al. [2001] included conjugate capability in an existing CFD code. The adjusted code was used to study conjugate heat transfer in an internal cooling configuration. The results were compared with the experimental data and were found in good agreement. Mazur et al. [2006] analyzed conjugate heat transfer of a gas turbine first stage nozzle. This helped in determining the metal maximum temperature and temperature profile, using cooling air flow rate and temperature. Kulasekharan et al. [2010] performed conjugate heat transfer calculations on cambered converged channels representing trailing edge of a gas turbine blade, with and without pins. They concluded that the effect of conjugation was very significant for pinned channels, especially for pin faces.
2.6 Summary of Chapter 2

From the literature survey on cooling methods used for gas turbine blades, it is found that the combination of internal and film cooling is used to keep the blade temperatures below the melting temperatures of the blade metal. Internal cooling at mid chord of the blades has been used extensively. Many heat transfer augmentation techniques have been studied which are available in open literature. This mainly includes the instalment of ribs of different configurations at the pressure and suction sides of the blade. Other methods like impingement cooling, pin-fin cooling and dimple cooling have also been studied. Steam has been found to have a good potential as a coolant in case of combined cycle power plants. The trailing edge has got a slot where from the coolant can bleed and leaves the blade’s inner cavity. With the usage of steam as a cooling medium, the blade metal temperatures can only be reduced with the help of internal cooling. To the knowledge of the author, the open literature does not provide any research done on internal cooling of trailing edge. The present study focuses on designing an appropriate internal cooling configuration targeting the trailing edge of a real blade. Conjugate heat transfer analysis is used with real engine boundary conditions.
3. METHODOLOGY

The method used in the current study is primarily numerical. Computational Fluid Dynamics (CFD) helps in the designing of cooling configurations in a gas turbine blade with predictions about the trends of the flow and heat transfer. The added advantage in using CFD is the 3D flow results compared to single measured points in experiments, which is useful in designing more efficient augmented heat transfer channels. In this thesis work, ANSYS FLUENT code was used as a CFD solver while ANSYS ICEM-CFD was used for mesh generation.

To fulfil the objectives of the thesis, the study was carried out in three steps. The three steps were:

1. CFD simulations of heat transfer and flow in two-pass smooth and ribbed rectangular channel which predicted the effect of reduction in the width of the inlet pass on heat transfer and pressure drop.

2. CFD simulations of heat transfer and flow in two-pass trapezoidal channel with three different cooling configurations for a trailing edge. The simulation results were used to compare the performance of the three proposed cooling configurations.

3. Conjugate heat transfer analysis with real blade boundary conditions and comparison of steam and air as coolants. The numerical results helped in designing an internal cooling configuration for the trailing edge that is suitable for limiting the maximum wall temperature below a certain value.

3.1 Rectangular Channel

Jenkins et al. [2008] experimentally studied the effect of variation in divider-to-tip wall distance in a two-pass rectangular channel with inlet pass having a different aspect ratio ($W_{in}/H = 1:2$) than the outlet pass ($W_{out}/H = 1:1$). To accommodate more channels in the chord-wise direction of the blade, the width of the inlet pass of the channel was reduced such that the aspect ratio of the inlet pass becomes equal to $W_{in}/H = 1:3$, in present thesis work. This is a unique combination which has never been studied before. The divider-to-tip wall distance was varied to get an optimized value with respect to thermal performance of the channel. The layout of the steps involved in rectangular channel simulation is shown in Figure 3-1.
For validation, the numerical results need to be compared with the experimental data. Therefore, numerical computations were started by modelling two-pass smooth rectangular channels with an inlet pass having a different aspect ratio ($W_{in}/H = 1:2$) than the outlet pass ($W_{out}/H = 1:1$). Experimental results by Jenkins et al. [2008] were used as the validation data. The effect of the reduction of the width of the inlet pass on heat transfer and pressure drop was studied by comparing the two types of the channels. To further extend the study, ribs at $45^\circ$ to the flow direction were added with rib height to hydraulic diameter ($c/D_h$) equal to 0.1 and pitch to rib height ($P/e$) equal to 10 in both channels. A similar comparison, as for the smooth channel, was again made for the ribbed channel. This study showed the importance of the variation in aspect ratio of inlet pass in two-pass channel in terms of heat transfer and pressure drop. The comparison of different aspect ratios of inlet pass of two-pass channels has never been studied before.

### 3.2 Trapezoidal Channel

The trailing edge of the gas turbine blade requires special attention particularly in the case where the restriction is to use internal cooling only. The trailing edge was modelled as trapezoidal channel. The layout of the steps involved in rectangular channel simulation is shown in Figure 3-2.
Experimental results by Lee et al. [2007] were used for validation of the numerical models. Different RANS turbulence models were compared for smooth as well as ribbed channels used in experiments by Lee et al. [2007]. The RANS models used included three version of k-ε model (low Re k-ε model, RNG k-ε model and realizable k-ε model) and two k-ω models (low Re k-ω model and SST k-ω model). The model which gave the lowest difference in results of heat transfer, in comparison with experiments, was selected for modelling the turbulence in the further designs of trailing edge heat transfer augmentation. Three different designs for internal cooling of the trailing edge of gas turbine blades were modelled and simulations were performed. These included:

A. Channel with ribs on trailing edge wall only.
B. Channel with ribs on trailing edge wall as well as on bottom wall, arranged in line to each other.
C. Channel with staggered ribs on trailing edge wall as well as on bottom wall.

The heat transfer augmentation and pressure drop offered by the three channels were compared at Reynolds number equal to 9400. This study helped in designing the trailing edge for fully internally cooled gas turbine blades. There are a very few studies on heat transfer and pressure drop in trapezoidal channels which makes a gap in the research. This thesis work contributes to the internal heat transfer augmentation in trapezoidal channels making it unique from previous work.

### 3.3 Conjugate Heat Transfer

Another numerical study was carried out which uses conjugate heat transfer analyses to design internal cooling passages for trailing edge of the gas turbine blades. It comprises of two sub-studies which are shown in the layout of the steps involved in conjugate heat transfer simulation, in Figure 3-3.

![Diagram of steps involved in trapezoidal channel simulation](image)

*Figure 3-2: Layout of the steps involved in trapezoidal channel simulation.*
Engine-similar boundary conditions were used to get temperature contours at different walls of the channel which represents the trailing edge of the blade. The internal configuration of the channel was altered with the aim of reducing the maximum metal temperature below the target value. A comparison was also made between steam and air as coolants. Traditionally the trailing edge is provided with trailing edge slots where from the coolant leaves the blade and mixes with the main gas flow. The current thesis work proposed a unique internal cooling channel design which can cool the trailing edge without letting the coolant to mix with the gas flow. Thus it reduces the aerothermal losses.

### 3.4 Overall

The overall study contributes in design process of augmented heat transfer in internally cooled gas turbine blade specially the trailing edge part. Contours of Nusselt number distribution at different walls of the channels were used to observe the enhancement in the heat transfer while area averaged Nusselt numbers were calculated to compare the heat transfer contributions for different designs. To compare the heat transfer enhancement and the associated pressure drop, a quantity called thermal performance was calculated for all the cases.
4. NUMERICAL MODELS AND VALIDATIONS

This chapter is dedicated to the description of the numerical models used for the analysis of different two-pass channels and the boundary conditions implemented. The computational results are validated against the available experimental measurement data. The description of the models and the boundary conditions applied in a conjugate heat transfer problem are also presented in the last section. The details of the governing equations and the turbulence modelling used in the CFD study in the thesis is described in Appendix A and B respectively.

4.1 Two-Pass Rectangular Channel

Two-pass channels are conventionally used to model the cooling passages inside a gas turbine blade. Basically two types of two-pass rectangular channels were studied namely smooth and ribbed channel. ANSYS ICEM-CFD was used to create geometries as well as to generate the structured mesh. ANSYS FLUENT was used to solve the numerical problem. This code uses the finite volume method to solve the governing equations describing the fluid flow and heat transfer under given boundary conditions. The realizable k-ε turbulence model with enhanced wall treatment was applied for turbulence modelling. Enhanced wall treatment is a method to model the near wall region that combines a two-layer model with enhanced wall functions [ANSYS FLUENT; 2009]. Shevchuk et al. [2011] has shown that this model performs well for separating flows behind an inclined rib at a Reynolds number of 100,000. When enhanced wall treatment is employed, y+ near 1 should be used but a value less than 4 to 5 is also acceptable [ANSYS FLUENT; 2009]. The near wall regions were meshed, such that the y+ value remains in the range of 1-3 for all cases which is a requirement of the near wall treatment used. The residuals for the continuity equation were set to reach $10^{-6}$ and that for the energy equation to $10^{-9}$, in order to assume the solution as converged.

The Nusselt number is based on the hydraulic diameter at the inlet of the channel. To study the effect of the bend and augmentation devices in a channel on heat transfer, it is necessary to compare it with the results for a straight smooth channel. Different experimental correlations are available in literature, which are used to calculate average Nusselt numbers in confined flows. The Dittus-Boelter correlation [Incropera et al.; 2002] has been used to normalize the Nusselt number in the current study and is defined as

$$Nu_o = 0.023 \ Re^{0.8} \ Pr^{0.4} \quad (4.1)$$

4.1.1 Boundary Conditions

At the inlet to the channel, the flow conditions are assumed to be fully developed. In order to apply a fully developed flow boundary condition, at the inlet of the two-pass channel, velocity, temperature and turbulence profiles were mapped from the outlet of a periodic segment to the
inlet of the two-pass channel. This periodic segment ensures that the flow is fully developed. The inlet mass flow rate for the periodic segment corresponds to the Reynolds number of 100,000, with the inlet temperature of 310K. Turbulence intensity of 3.8% was used at the inlet. This was calculated by the relationship provided by ANSYS Fluent [2009] shown as Eq. 4.2.

\[ I = 0.16 \text{Re}^{-1/8} \]  \hspace{1cm} (4.2)

All walls were kept at a constant temperature of 350K. Symmetry was introduced at the top of the computational domain shown in Figure 4-1, which reduces the computational effort. Ambient conditions were set at the outlet.

The flow is considered incompressible, three dimensional, turbulent and steady with constant thermodynamic properties. The working fluid is dry air with Prandtl number equal to 0.71.

The grid independence study and validations of numerical results for both types of channels are presented separately.

4.1.2 Two-Pass Smooth Rectangular Channel

Description of the Physical Model

A schematic of the geometrical model of the smooth two-pass channel used in the study is shown in Fig. 4-1. Due to symmetry, only half of the channel is simulated.

*Figure 4-1: Schematic view of the smooth two-pass channel with \( W_{in}/H = 1:3 \) at inlet pass and \( W_{out}/H = 1:1 \) at outlet pass.*
The inlet pass has an aspect ratio \((AR_{in} = \frac{W_{in}}{H})\) of 1:3, while the outlet pass is a square channel i.e., \((AR_{out} = \frac{W_{out}}{H})\) of 1:1. These two are connected by a 180° bend. The channel height is \(H = 150\) mm, while the inlet width is \(W_{in} = 50\) mm. The hydraulic diameter of the inlet pass is 75 mm. A divider wall of thickness \(W_{web} = 20\) mm separates the two passes. The edge of this wall has been rounded in the bend region with a curvature radius of 10 mm. The divider-to-tip wall distance \((W_{el})\) was varied from 50 mm to 150 mm in order to study its effect on the flow and heat transfer.

**Grid Independence**

A detailed grid independence study was carried out where three progressively finer grids were compared, for a fixed geometry with \(W_{el} = 75\) mm \((\frac{W_{el}}{W_{in}} = 1.5)\). The three cases namely Case 1, Case 2 and Case 3 comprise of grid sizes equal to 950K, 1.7M and 2.6M cells respectively. Figure 4-2 shows the bend region of the three meshes. The near wall mesh needed for modelling the viscous sub layer was kept constant so that \(y^+\) at the wall is maintained \(\sim 1\) and the growth from the wall is at a ratio of 1.2.

![Figure 4-2: Three grids at the bend section of the channel with \(W_{el}/W_{in} = 1.5\).](image)

The averaged Nusselt number at the bottom of the inlet, bend and outlet regions were compared. Figure 4-3 shows the comparison for the three cases. The area averaged Nusselt number in each region of the channel has been normalized with the Nusselt number \((\text{Nu}_{\text{st, avg}})\) obtained from a smooth periodic segment. All three cases produced similar results with the maximum deviation
of 0.88%. Keeping in mind that the acceptable results should be obtained in a computationally economical way, the case with coarse grid was chosen for the further simulations.

**Figure 4-3:** Comparison of the average Nusselt numbers in different regions for the three cases of the smooth two-pass channel with $W_{el}/W_{in} = 1.5$.

**Validation**

Computational Fluid Dynamics (CFD) has shown to be adequate for predicting the heat transfer trends in channels. Over the years extensive work has been done in its development and improvements in its ability to predict. But still a comparison between numerical simulations and experiments shows differences, which can be due to inadequate numerical modelling or measurement errors in the experiments. A turbulence model may be very good in predicting the desired results for one case, but might fail in doing so in another case. In practice today, the CFD results for a certain case are compared to experimental results and then, if found good, the numerical results of other similar cases are considered as accepted.

Jenkins et al. [2008] performed experiments at Reynolds number equal to 100,000 to study the effect of the tip wall distance on heat transfer for a varying aspect ratio two-pass channel which includes a $W_{in}/H = 1:2$ inlet and a $W_{out}/H = 1:1$ outlet aspect ratio. These inlet and outlet channels were connected with a 180° bend. The results for smooth channel with divider wall-to-tip wall distance ($W_{el}/W_{in}$) equals to 1.5 were selected as validation case in the current study. Figure 4-4
shows the local Nusselt numbers at different locations in the channel normalized with the Nusselt number as defined in Eq. 4.1. Experiments showed that the flow gets fully developed when it reaches near the bend region, so a periodic segment was used to produce the fully developed profiles which were mapped at the inlet to the computational domain shown in Fig. 4-1.

![Figure 4-4: Local Nusselt number distribution by experimental (left) and numerical (right) results.](image)

CFD predictions of the local Nusselt number distribution are in good agreement with the experimental results at inlet and bend bottom regions (1 and 3 respectively). At the tip wall (4), the experiments show two regions of high heat transfer separated by a lower heat transfer region. The numerical results, however predict a larger region of enhanced heat transfer due to impingement of fluid coming from the inlet pass. At the side wall (2) in bend region the numerical prediction is very similar to the experimental results though a comparatively larger region with high heat transfer is predicted. At the side walls of the bend (5) and the outlet pass region (6), the numerical results are also comparable to the experimental results except for the bottom surface of the outlet pass (6), where a large recirculation region is predicted which results in reduced heat transfer close to the divider wall and increased heat transfer at the side walls.
possibly a result of a vena contracta in flow. But the experimental results suggest a less severe separation at the 180° bend.

The area averaged Nusselt numbers for different regions were also compared with experimental results and are shown in Table 4-1. At the tip wall, due to a prediction of higher impingement, the difference in numerical and experimental values is high. Though the local Nusselt number distribution at the outlet pass was different, the area averaged values are not that different. This means that the prediction of more abrupt change in Nusselt number has minor effect on the area average value in this case.

Table 4-1: Comparison of average Nusselt numbers from experiments and numerical simulation for $W_{el}/W_{in} = 1.5$.

<table>
<thead>
<tr>
<th>Region</th>
<th>$\frac{Nu_{avg}}{Nu_0}$ (Numerical)</th>
<th>$\frac{Nu_{avg}}{Nu_0}$ (Experimental)</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Inlet)</td>
<td>0.80</td>
<td>0.82</td>
<td>-2.4%</td>
</tr>
<tr>
<td>2 (Bend side wall near inlet pass)</td>
<td>1.17</td>
<td>1.07</td>
<td>9.3%</td>
</tr>
<tr>
<td>3 (Bend bottom)</td>
<td>1.85</td>
<td>1.73</td>
<td>6.9%</td>
</tr>
<tr>
<td>4 (Tip wall)</td>
<td>1.91</td>
<td>1.59</td>
<td>20.1%</td>
</tr>
<tr>
<td>5 (Bend side wall near outlet pass)</td>
<td>2.21</td>
<td>1.96</td>
<td>12.7%</td>
</tr>
<tr>
<td>6 (Outlet)</td>
<td>1.44</td>
<td>1.37</td>
<td>5.1%</td>
</tr>
</tbody>
</table>

RANS (Reynolds Averaged Navier Stokes) models are known to perform somewhat worse for the flows with strong anisotropy of turbulence, for instance, in the bend area. A remedy would be to use a LES or a DES model of turbulence, which are however very time-consuming. In view of the fact that the overall performance of the selected turbulence model in the inlet and outlet passes and to some extent in the bend area is good, and that it provides fast and robust convergence of the simulations, these advantages overweighed the decreased accuracy in the bend area.

Shevchuk et al. [2011] also performed numerical simulations for flow and heat transfer in the identical channel. The results were validated against experimental results by Jenkins et al. [2008]. Shevchuk et al. [2011] used an unstructured mesh for their numerical simulations, whereas a structured mesh has been used in the current study. A structured grid allows for more
control over the mesh parameters e.g., mesh size near the wall. Also the grid is of good quality with little cost on CPU time as it is generated faster than the unstructured mesh. The average Nusselt number for different channel sections at different divider-to-tip wall distances and their comparison with Shevchuk et al. [2011] is shown in Fig. 4-5. Both simulations produced comparable results to the experiments, with maximum difference of near 8% at outlet pass for \( W_{el}/W_{in} \) equal to 1. Thus the choice of mesh type depends mainly upon the available resources and ease to handle.

![Figure 4-5: Effect of \( W_{el} \) on average heat transfer in the inlet pass (faces 1a and 2), bend (face 3 and 4) and outlet pass (faces 6a and 5) as predicted in current simulation and by Shevchuk et al [2011]](image)

4.1.3 Two-Pass Ribbed Rectangular Channel

**Description of the Physical Model**

A scheme of the geometrical model of the ribbed two-pass channel used in the study is shown in Fig. 4-6. The inlet pass has aspect ratio (\( AR_{in} = W_{in}/H \)) of 1:3, while the outlet pass is a square channel with \( AR_{out} = W_{out}/H = 1:1 \). These two channels are connected with a 180° bend. The channel height is 150 mm, while the inlet width is 50 mm. The hydraulic diameter of the inlet pass is 75 mm. A divider wall of thickness \( W_{web} = 20 \) mm separates the two passes. The edge of the divider wall has been rounded in the bend region with a curvature radius of 10 mm. The divider-to-tip wall distance (\( W_{el} \)) was varied from 75 mm to 150 mm in order to study its effect on the flow. The rib height to hydraulic diameter ratio (\( e/D_h \)) was kept constant to a value of 0.1 for both inlet and outlet passes. Similarly the pitch-to-rib height distance (\( P/e \)) was kept equal to 10. The ribs were installed inline on both top and bottom surfaces thus allowing half of the channel to be modelled in simulations with the introduction of a symmetry boundary condition at
the upper surface of the computational domain. To avoid back flow effect, the channel was extended to a length equal to hydraulic diameter of the outlet pass.

**Figure 4-6:** Schematic view of the ribbed two-pass channel with $W_{in}/H = 1:3$ in the inlet pass and $W_{out}/H = 1:1$ at outlet pass and a symmetry boundary condition on the upper surface.

**Grid Independence**

A detailed grid independence study was carried out for the ribbed channel. Three grid sizes were compared, where $W_{el}/H = 0.88$. The three cases, namely Case 1, Case 2 and Case 3 comprise the grid sizes equal to 3.5, 4.6 and 5.7 million cells, respectively. Figure 4-7 shows a section of these grids. The first cell near the walls was kept such that the y+ value is near 1. The growth rate of the mesh from the wall was kept constant to 1.2.

**Figure 4-7:** A section of the three grids used for grid independence study.
For the grid independence study, the averaged normalized Nusselt numbers \((\overline{Nu}/Nu_o)\) for different regions of the channel were compared. Figure 4-8 shows the results for these locations which are identified in Fig. 4-6. Note that the combined average of face 5, 6 and 7 does not include the rib area. Other than face 3, where the difference of Case 2 was found to be 4 and 5% from Case 3 and Case 1 respectively, all values for Case 2 were found comparable to Case 3. Due to computational economy Case 2 was selected for further analyses. The same grid system was then used for all other cases of the \(W_{el}/H\) variation.

**Figure 4-8:** Comparison of the average Nusselt numbers normalized with Dittus-Boelter correlation in different regions for the three cases.

**Validation**

Jenkins et al. [2008], performed experiments at Reynolds number equal to 100,000 to study the flow and heat transfer in two-pass internal ribbed cooling channel, which includes a 1:2 inlet and 1:1 outlet aspect ratio (W/H). These inlet and outlet channels were connected with a 180° bend. Multiple geometries were studied with \(W_{el}\) ranging from 75 mm to 150 mm. The case with \(W_{el}\) equal to 121.5 mm was selected for validation. Figure 4-9 shows the local distribution of Nusselt numbers normalized with the smooth channel value \((Nu/Nu_o)\) for both the experiment (left) as well as numerical validation (right).
The numerical solution’s predictions at the bend and at the inlet to the outlet pass are overall good. The separation bubble after the turn is predicted well by the numerical simulation. At the tip (4) the predictions are again acceptable as a hot spot here is captured though low heat transfer is predicted near the walls. At the region between first and second rib (5) in the outlet pass, the predictions are again good enough and a region of high heat transfer after the rib is captured. Similarly at regions 6 and side wall (8) the predictions are acceptable. It was concluded that the model is fairly good in predicting the local heat transfer values.

The area averaged Nusselt numbers are the other way of judging the validation of numerical results. It was found that except at the bend and the tip regions the area averaged Nusselt numbers differs about 12% from the experimental results. At the bend and tip this difference was about 27%. The very low heat transfer values present locally at these regions contributes to this difference. The numerical techniques include DNS (Direct Numerical Simulations), LES (Large Eddy Simulations) and Reynolds Averaged Navier Strokes (RANS) models, mainly. Although accurate results have been obtained with help of DNS and LES but their dependency on grid size makes them very expensive in case of a complete two-pass channel. So, considering the fast and robust convergence of the simulations using RANS, somewhat decreased accuracy in the bend area can be tolerated ([Swell et al.; 2005] and [Viswanath et al.; 2006]). Okamura et al. [2002] compared the experimental measured Nusselt numbers with computed ones for a high aspect...
ratio rectangular channel with ribs at 45°. It was found that the ratio between the highest and lowest heat transfer coefficient in the ribbed wall was about 2.5 in the calculation while the measured ratio was about 3.0 making a difference of about 17%. Shih et al. [2001] found a peak difference of about 20% while comparing experimental data and computed results for heat transfer in a non-rotating two-pass channel with ribs at two walls. Su et al. [2004] computed flow and heat transfer in a two-pass smooth channel (with rotation as well as non-rotation) with RANS equations in conjunction with a near-wall Reynolds stress turbulence model. The difference of the computed Nusselt numbers with experimental data, at the inlet pass for non-rotating channel were found to be as high as 19% while at bend the maximum difference was found to be 38%. Comparing to this example from open literature, the difference of 27% between experimental and numerical results at the bend is acceptable. Similarly at inlet pass a difference of 12% is also in acceptable range if compared to Su et al. [2004]. Also, in present study the averaged Nusselt numbers at different sections of the channel are within similar accuracy to Shevchuk et al [2011], Su et al [2004], Okamura et al. [2002] and Shih et al. [2001]. Thus it can be concluded that the model is also fairly good in predicting the area averaged heat transfer values.

4.2 Two-Pass Trapezoidal Channel

A trapezoidal channel is used for modelling the cooling passages at the trailing edge of the gas turbine blade. In the present study, three different configurations of two-pass trapezoidal channels have been studied focusing the trailing edge. Table 4-2 shows the details of the three configurations.

<table>
<thead>
<tr>
<th>Case</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case A</td>
<td>Ribs on trailing edge only</td>
</tr>
<tr>
<td>Case B</td>
<td>Inline ribs on trailing edge and bottom wall of the outlet pass</td>
</tr>
<tr>
<td>Case C</td>
<td>Staggered arrangement of ribs at trailing edge and the bottom wall of the outlet pass</td>
</tr>
</tbody>
</table>

To create geometries and meshes, ANSYS ICEM-CFD was used. ANSYS FLUENT was used to solve the numerical problem. As the flow in the two-pass ribbed channels experience phenomenon like flow mixing, correct prediction of the flow field and heat transfer depends on the choice of turbulence model used. As discussed earlier, Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) not only require very refined mesh but are also time consuming. This makes these unpopular in terms of computational cost economics. Reynolds averaging of the Navier-Stokes (RANS) equations is thus an acceptable method if the choice of the turbulent model is made correctly. Lee et al. [2007] performed experiments to study the heat transfer distribution in a two-pass trapezoidal channel. For validation of numerical results, the
experimental results by Lee et al. [2007] for a smooth channel and a channel with ribs at bottom wall, with flow inlet at a larger cross-section were selected. Five different turbulence models provided by ANSYS FLUENT were tested. These include three k-ε models (low-Re k-ε model, RNG k-ε model and realizable k-ε model) and two k-ω models (low-Re k-ω model and SST k-ω model). To model the near-wall region, enhanced wall treatment was used. The enhanced wall treatment combines the two-layer model with enhanced wall functions. The y+ value near the wall should be close to 1 but values less than 4 to 5 are also acceptable [ANSYS FLUENT; [2009]. In all cases, the near-wall regions were meshed such that the y+ value remain in the range of 1 to 2. The values of the Nusselt number at the bottom wall were monitored and the convergence was assumed when these values cease to change.

4.2.1 Description of the Physical Model

Three designs for augmented heat transfer at the trailing edge of the gas turbine blade have been analyzed. Figure 4-10 shows the schematic of the geometrical model of these designs. The basic geometric parameters are the same as used by Lee et al. [2007]. A 571 mm long inlet pass is connected to an outlet pass with 180° bend. The divider-to-tip wall distance is equal to the width of the inlet channel (W = 38.1 mm), while the thickness of the divider wall is 19.1 mm. The outlet pass extended after the last rib to avoid reverse flow effects is 305 mm long. The maximum height of the inlet pass (H₁) is 76.2 mm, while the minimum height of the outlet pass (H₂) is 38.1 mm. This makes the angle between the top and bottom wall equal to 21.8 deg. The ribs, which are orthogonal to the flow direction, are placed at a distance of 305 mm from the inlet. The square cross-sectioned ribs have height of e = 3.2 mm. The pitch-to-rib height distance (P/e) is equal to 12.
All the three cases have the similar geometric configuration at inlet pass, therefore in Fig. 4-10 the inlet pass is only shown for one case. In Case A, there are no ribs at the bottom surface of outlet pass and the trailing edge is provided with ribs of the same dimensions as those at the inlet pass. Case B, has ribs at both bottom as well as at the trailing edge. The ribs at the bottom wall are in line with the ribs at the trailing edge. A staggered arrangement of ribs at the outlet pass has been analyzed in Case C.

4.2.2 Boundary Conditions

All the calculations were done at the Reynolds number equal to 9400, which is defined at the turn of the channel. The mass flow rate calculated from Eq. 4.3 and was used as an inlet boundary condition.

\[
\text{Re} = \frac{\rho V D_h}{\mu} = \frac{4m}{\mu Pe} \tag{4.3}
\]
Turbulence intensity of 5% was used at the inlet. This was calculated by the relationship provided by ANSYS FLUENT [2009] shown as Eq. 4.2. In their experiments Lee et al. [2007] used naphthalene sublimation technique to measure mass transfer and then used an analogy to estimate the heat transfer, taking into account the fact that the constant temperature wall boundary condition is analogous to the constant wall concentration of naphthalene in the experiments. The wall was considered to be at 350K and the fluid inlet temperature was selected to be 310K. For outlet pass the length was selected equal to the length on which experimental data was taken and an extension equal to twice the hydraulic diameter of the bend region was introduced to take care of reverse flow. Outflow boundary condition was imposed at the outlet.

4.2.3 Grid Independence

ANSYS ICEM-CFD is used to generate structured meshes. Figure 4-11 shows a section of the three grids used for the grid independence study of the most complex case i.e., the staggered ribbed channel.

![Figure 4-11: Three grids used for grid independence study.](image)

The three grids namely coarse, fine and finest represent grid sizes equal to 1.9 million, 3.1 million and 4.8 million cells respectively. The near-wall mesh was kept constant in all three cases so that y+ immediately near the wall was maintained close to unity, and the growth from the wall is at a ratio of 1.2. Area averaged Nusselt numbers at different segments of the channels were compared for the three grids. The details of these segments are in the next section. A maximum of 1.85 % deviation in the area averaged Nusselt number was found between the coarsest and finest mesh. To keep the balance between numerical accuracy and computational economy, the coarse grid was selected for the further analysis. The same meshing strategy was adopted for the other channels too.
4.2.4 Validation

Lee et al. [2007] performed experiments to study the heat transfer in a trapezoidal channel. The results for the Nusselt number in the smooth channel and the channel with ribs at the bottom wall at the Reynolds number of 9400 have been selected for validation of the numerical model. For the smooth channel, three k-ε models (low-Re k-ε model, RNG k-ε model and realizable k-ε model) and two k-ω models (low-Re k-ω model and SST k-ω model) were evaluated. All these turbulence models and their empirical constants were used as they are implemented in the ANSYS FLUENT software. The channel was divided into 16 segments by Lee et al. [2007]. For validation purposes, the same segment numbers were used as by Lee et al. [2007] and are shown in Fig 4-12.

Figure 4-12: Segment numbers at the bottom wall of the channel.

Figure 4-13 shows the comparison of k-ε models with experimental results for the smooth channel. All the models were successful in predicting the true trends at the inlet and outlet pass but were unable to catch the peak value at the band. All these models under predict the Nusselt numbers at the inlet pass (till section 7) and bend region (8-9) with the difference of about -10 to -24%. At the outlet pass (10-16) the low-Re k-ε model predicts better than the other models with an average difference of about +7%.

Figure 4-13: Comparison of numerical results from k-ε models and experimental data for the smooth channel.
Figure 4-14 shows the comparison of two $k-\omega$ models with experimental data for the smooth channel. Both versions of the model predict the same trends with almost similar difference ranges but the peak value was predicted at the start of outlet pass instead of experimental peak value at the bend. At the inlet the average difference was about -16% for both cases. At the bend, SST $k-\omega$ model performs better with difference of -12% in contrast to the low-Re $k-\omega$ model, which resulted in difference of about -20%. At the outlet pass, the average difference was found lowest in case of the SST $k-\omega$ model, that was +15% away in comparison with +18% of the low-Re $k-\omega$ model.

![Figure 4-14: Comparison of numerical results from $k-\omega$ models and experimental data for the smooth channel.](image)

From each of the two groups ($k-\varepsilon$ and $k-\omega$) one turbulence model with the least differences in prediction of heat transfer in the two-pass smooth trapezoidal channel was selected. The low-Re $k-\varepsilon$ model and SST $k-\omega$ model were selected to predict heat transfer in the ribbed two-pass trapezoidal channel with geometrical configurations proposed by Lee et al. [2007]. Figure 4-15 shows the comparison of these two models with experiments for the ribbed channel. The SST $k-\omega$ model performs the worst and under-predicts Nusselt numbers in the inlet pass with an average difference of about -60% in contrast to the low-Re $k-\varepsilon$ model which performs better, comparatively with an average difference of -34%. At the inlet the flow was not fully developed, therefore this under-prediction may have resulted from the fact that the two-equation models are suitable for fully developed turbulent flows. At the bend, the SST $k-\omega$ model performs better as could be seen in previous investigations. The average difference at the bend with this model was found to be about -3% in comparison to -9% by the low-Re $k-\varepsilon$ model. Also, experiments found the peak of the Nusselt number upstream of the outlet pass (segment 10), which was captured by both the models though in magnitude the low-Re $k-\varepsilon$ model was found much closer to the experimental value. The experiment also showed that the peak value remained constant in the
subsequent segment. However both turbulence models failed to capture that. In the outlet pass, the average difference resulted from use of the low-Re $k-\varepsilon$ model was approximately -17%, whilst the SST $k-\omega$ model entailed -33%.

![Figure 4-15: Comparison of numerical results from the low-Re $k-\varepsilon$ model and SST $k-\omega$ model with experimental data for the ribbed channel.](image)

Lee et al. [2007] calculated pressure losses in the form of the friction factor ratio defined by Eq. 4.3.

$$f = 2\rho \left( \frac{A_p}{L} \left( \frac{A}{m} \right)^2 \right) D_h$$

(4.3)

The friction factor is normalized by the friction factor for fully developed turbulent flow in a smooth channel, which is defined as

$$f_o = \left[0.790 \ln(Re) - 1.64\right]^2$$

(4.4)

Table 4-3 presents the experimental and numerical results for both smooth and ribbed channels using the low-Re $k-\varepsilon$ model.
Table 4-3: Pressure losses in the smooth and ribbed trapezoidal channels.

<table>
<thead>
<tr>
<th></th>
<th>$f/f_0$ (Exp)</th>
<th>$f/f_0$ (CFD)</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Smooth</td>
<td>9.3</td>
<td>11.7</td>
<td>+25 %</td>
</tr>
<tr>
<td>Ribbed</td>
<td>13.7</td>
<td>13.5</td>
<td>−1.5%</td>
</tr>
</tbody>
</table>

The aim of the study is to evaluate different configurations of the trailing edge, therefore more emphasis is made on matching the numerical predictions with experimental data at the outlet pass. The trends of heat transfer distribution at different sections of the outlet pass of the channel are predicted successfully. The experimental data has an uncertainty of about ±8% in the Nusselt number and ±7.8% in the friction factor. The predictions by the low-Re k-ε turbulence model are thus acceptable for the further analysis of the other configurations of the two-pass channel as it has the minimum difference from experimental data in the outlet pass.

4.3 Conjugate Heat Transfer Analysis

Conventionally, the coolant (air) is fed from compressor which is allowed to leave the blade through the film cooling holes and trailing edge slots. This on one hand protects the blade from high temperatures of the gas but on the other hand it results in reducing its temperature and also increases aerodynamic losses. Also with the use of steam as coolant, it is not allowed to leave the blade through film holes or trailing edge slots. Thus to avoid this, a better design of internal cooling of trailing edge is required. This study focuses on the design of a two-pass trapezoidal channel, which represents the trailing edge of the vane of a real engine. The real engine conditions are applied at the walls. The flow rates used correspond to a Reynolds number of 100,000 based on the hydraulic diameter at inlet. The challenge is to design a two-pass trapezoidal channel which is capable to reduce the metal temperatures below a target value with minimal pressure drop. The coolant to be used is steam with a fixed given thermal conditions. The challenge is approached by initially designing a smooth two-pass trapezoidal channel and then altering its design by focusing on different effects of flow inside the channel.

Siddique et al. [2011] has shown that the low-Re k-ε turbulence model with enhanced wall treatment performs better for predicting heat transfer and pressure drop in trapezoidal channel compared to other RANS models. Thus low-Re k-ε turbulence model with enhanced wall treatment was applied for turbulence modelling. Enhanced wall treatment is a method to model the near wall region that combines a two-layer model with enhanced wall functions [ANSYS FLUENT; 2009]. When enhanced wall treatment is employed, $y^+$ near 1 should be used but a value less than 4 to 5 is also acceptable [ANSYS FLUENT; 2009]. The near wall regions were meshed, such that the $y^+$ value remains in the range of 1-3 for all cases which is requirement of the near wall treatment used. The growth rate of the mesh from the wall was kept constant to 1.2. The residuals for continuity equation were allowed to reach $10^{-6}$ and that for energy equation to $10^{-9}$, in order to assume the solution as converged.
4.3.1 Description of the Physical Model

The design process was initiated by studying the effect of heat transfer and pressure drop in a smooth two-pass trapezoidal channel. Eight different variations in that base case were then studied. A brief description of these cases is provided in Table 4-4.

Table 4-4: Different two-pass channel configurations studied.

<table>
<thead>
<tr>
<th>Case</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case A</td>
<td>Two-Pass Trapezoidal Smooth Channel</td>
</tr>
<tr>
<td>Case B</td>
<td>Two-Pass Trapezoidal Ribbed Channel</td>
</tr>
<tr>
<td>Case C</td>
<td>Two-Pass Trapezoidal Channel with Ribs at Trailing Edge only</td>
</tr>
<tr>
<td>Case D</td>
<td>Two-Pass Trapezoidal Channel with Ribs at Trailing Edge as well as at Bottom Wall (Inline)</td>
</tr>
<tr>
<td>Case E</td>
<td>Two-Pass Trapezoidal Channel with Ribs at Trailing Edge as well as at Bottom Wall (Staggered ribbed)</td>
</tr>
<tr>
<td>Case F</td>
<td>Two-Pass Trapezoidal Smooth Channel with Fin at Trailing Edge (small)</td>
</tr>
<tr>
<td>Case G</td>
<td>Two-Pass Trapezoidal Smooth Channel with Fin at Trailing Edge (medium)</td>
</tr>
<tr>
<td>Case H</td>
<td>Two-Pass Trapezoidal Smooth Channel with Fin at Trailing Edge (large)</td>
</tr>
<tr>
<td>Case I</td>
<td>Two-Pass Trapezoidal Ribbed Channel with Fin at Trailing Edge</td>
</tr>
</tbody>
</table>

Figure 4-16 shows the schematic of the two-pass smooth channel (Case A), which is the baseline case. The length, L, of all the channels is equal to 75 mm. The total external width of the channel at the middle is \( W_{T,s} = 15 \) mm while the total internal width of the channel is \( W_{T,f} = 11.2 \) mm. The thickness of the channel at side wall and tip wall is \( t = 0.8 \) mm. The channel is divided into two passes, such that the width of the inlet pass is \( W_{In} = 2.5 \) mm while that of the outlet pass is \( W_{Out} = 7.9 \) mm. The two passes are separated by a 0.8 mm thick divider wall. The gap between the divider wall and tip wall is 3.7 mm. The channels are trapezoidal so the height changes (reduces) from side wall to the trailing edge. The maximum external height of the channel is at the side wall which is equal to \( H_{In,s} = 7.6 \) mm while the maximum internal height of the channel is \( H_{In,f} = 5.5 \) mm, at the inlet pass. The height of the other side of the inlet pass is 4.4 mm. Similarly the maximum internal height at the outlet pass is \( H_{Out,f} = 4 \) mm while the other side is 0.6 mm high. The trailing edge height at the external side is \( H_{TE} = 1 \) mm. Due to symmetric geometry configurations, only half of the channel has been modelled in simulations with the introduction of a symmetry boundary condition at the upper surface of the computational domain. The width of the trailing edge wall at this symmetric plane is \( W_{TE} = 3 \) mm.
It is assumed that the hottest region of the channel would be the trailing edge wall. If that region is cooled down somehow to the target value, the channel design would be approved as acceptable. Thus the initial variations were made in the design of the outlet pass, all having the same geometric design at inlet pass.

Figure 4-17 shows the schematic of the outlet-pass of the two-pass ribbed channel (Case B). In this case the bottom walls of the outlet pass are roughened with 45° ribs such that the distance between ribs (pitch) is $P = 3.5\, \text{mm}$. A total of 18 ribs were installed at the outlet pass. These are square sectioned ribs with height at one end (attached to divider wall) equal to $e_1 = 0.35\, \text{mm}$ while at other end (attached to trailing edge) it is equal to $e_2 = 0.16\, \text{mm}$ thus making them trapezoidal along the width.
Figure 4-17: Schematic view of the two-pass ribbed channel (Case B). Inset is the top view of a section of the same channel.

Figure 4-18 shows the schematic of the outlet pass of the channel with ribs at the inner side of the trailing edge only (Case C). The rib pitch equal to $P = 3.5$ mm and the width of the rib is equal to $e_2 = 0.16$ mm which is also equal to the height of the rib, $r_1$.

Figure 4-18: Schematic view of the two-pass channel with ribs at the trailing edge only (Case C). Inset is the close up of the same channel.
Figure 4-19 shows the schematic of the outlet pass of the channel with ribs at the inner side of the trailing edge as well as at the bottom wall inline to each other (Case D). The pitch of the ribs is equal to $P = 3.5$ mm. The ribs configuration is the same as for Case B and C.

![Figure 4-19: Schematic view of the two-pass channel with ribs at the trailing edge as well as at the bottom wall with inline arrangement (Case D). Inset is the close up of the same channel.](image1)

Figure 4-20 shows the schematic of the outlet pass of the channel with ribs at the inner side of the trailing edge as well as at the bottom wall in staggered arrangement (Case E). The pitch of the ribs is equal to $P = 3.5$ mm for both at bottom wall as well as at the trailing edge. The ribs configuration is the same as for Case B and C.

![Figure 4-20: Schematic view of the two-pass channel with ribs at the trailing edge as well as at the bottom wall with staggered arrangement (Case E). Inset is the close up of the same channel.](image2)
Figure 4-21 shows the front view of the two-pass smooth trapezoidal channel with a fin at the trailing edge. Fin width, $W_{\text{Fin}}$ is varied from small size, medium and large size making three different cases. For Case F, the total width of the fin at the symmetric plane is equal to 3.4 mm while for Case G and Case H it is equal to 3.8 mm and 4.2 mm respectively.

![Figure 4-21: Front view of the two-pass smooth channel with a small fin at the trailing edge (Case F, G and H).](image)

The front view of the two-pass ribbed trapezoidal channel with fin at the trailing edge (Case I) is shown in Figure 4-22. The total width of the fin at the symmetric plane is equal to that of Case F (3.2 mm). The difference is that it is provided with ribs at bottom wall of outlet pass. The rib configuration at side wall is same as that of Case B but it is truncated near the trailing edge wall. It is done to allow fluid to flow more smoothly near the trailing edge. This results in a square cross sectioned rib near the trailing edge wall with height and width equal to 0.17 mm.

![Figure 4-22: Front view of the two-pass ribbed channel with fin at the trailing edge (Case I).](image)

### 4.3.2 Boundary Conditions

For a general conjugate heat transfer problem, the flow conditions of the inner as well as outer flows are known and these are used to calculate the heat transfer coefficients at the inner and outer surfaces of the geometric domain. In the current study, the inner flow conditions are known while at the outer surfaces of the channel; heat transfer coefficients as well as the bulk temperature of the hot gas are known. Table 4-5 gives the heat transfer coefficients and the hot gas bulk temperatures. All other walls are kept adiabatic.
Table 4-5: Wall boundary conditions applied to all cases.

<table>
<thead>
<tr>
<th>Wall</th>
<th>Heat transfer coefficient (W/m² K)</th>
<th>Bulk temperature of hot gas (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Side Wall</td>
<td>1770</td>
<td>653</td>
</tr>
<tr>
<td>Trailing Edge</td>
<td>2777</td>
<td>1374</td>
</tr>
<tr>
<td>Bottom Wall</td>
<td>2950</td>
<td>1374</td>
</tr>
<tr>
<td>Tip Wall</td>
<td>3000</td>
<td>1374</td>
</tr>
</tbody>
</table>

At the inlet of the channel, Reynolds number is 100,000. The corresponding mass flow rate at the inlet is used as boundary condition while the temperature of the fluid is 653K. Symmetry is introduced at the top of the computational domain, which helped in reducing the computational effort. The flow is considered as incompressible, three dimensional, turbulent and steady with constant thermal physical properties. The working fluid is superheated steam with degree of superheat equal to 204K and absolute pressure equal to 905 kPa. The Prandtl number of the fluid under these conditions is 0.94.

4.3.3 Design Approach

The current study is focusing on the design of a two-pass trapezoidal channel which by use of internal cooling can reduce the wall temperatures below a target value with minimal pressure drop. The temperatures (T) at the walls of the channel have been normalized with the target temperature value (T_{targ}). The approach is to achieve a design of the two-pass trapezoidal channel which has a value T/T_{targ} less than 1 for every region of the channel walls. As the hottest region is the trailing edge therefore data for trailing edge corner, shown in Fig. 4-16, have been plotted for comparison of different designs.

4.3.4 Grid Independence

A grid independence study was carried out for smooth (Case A) and ribbed channels (Case B). For the smooth channel, three grids sizes were tested for grid independence. The three cases namely Case 1, Case 2 and Case 3 comprises of grid sizes equal to 0.8M, 1.2M and 1.6M cells respectively. Normalized temperature profiles at the trailing edge corner, as mentioned in Fig. 4-16, were compared for the three cases. These profiles are shown in Fig. 4-23. Very near the turn (Z/Z_{max} = 0 to 0.1) there exists a difference of about 0.5% in normalized temperature between the Case 2 and Case 1 as well as Case 3. This difference is less further along the length of the channel. Thus Case 1 was selected for further analysis. The same grid sizing was also selected for other design of the two-pass trapezoidal channels with smooth walls but with fins i.e., Cases F, G and H.
For the ribbed channel, three grids were tested for grid independence. The three cases namely Case 1, Case 2 and Case 3 comprises of grid sizes equal to 3M, 4M and 5.4M cells respectively. Normalized temperature profiles at the trailing edge corner were compared for the three cases. These profiles are shown in Fig. 4-24.

The normalized temperature values are very similar for Case 2 and 3 all along the line. Keeping in mind that the acceptable results should be obtained in a computationally economical way,
Case 2 was chosen for the further simulations. The same grid sizing was also selected for other design of the two-pass trapezoidal channels with ribs i.e., Cases C, D, E and I.
5. RESULTS FROM RECTANGULAR CHANNEL SIMULATIONS

In chapter 4, the physical models and the validation of numerical methods for two-pass rectangular channels (smooth and ribbed) have been presented. The divider-to-tip wall distance has been varied for the channel with an inlet channel aspect ratio \( W_{in}/H = 1:3 \) and its effect on heat transfer on different sections of the channel has been calculated along with the associated pressure drop. The results are then compared to those of a channel with the inlet channel aspect ratio \( W_{in}/H = 1:2 \). For all cases, the outlet channel aspect ratio was kept constant at \( W_{out}/H = 1:1 \). The results obtained for smooth and ribbed channel are presented separately in the sections below. The CFD simulation of two-pass smooth and ribbed channels with inlet pass aspect ratio of 1:2 has been validated against experimental results of Jenkins et al. [2008] and also compared with CFD results of Shevchuk et al. [2011] and has been presented in Chapter 4. This numerical model is therefore accepted as a benchmark to study heat transfer and flow in a two-pass smooth as well as ribbed channel with inlet pass aspect ratio of 1:3.

5.1 Two-Pass Smooth Rectangular Channel

5.1.1 Pressure Drop and Heat Transfer

To observe the effect of varying the divider-to-tip wall distance on pressure drop, the relative pressure drop \( \Delta p^* \) has been calculated using Eq. 5.1.

\[
\Delta p^* = \frac{\Delta p}{0.5 \rho U_{in}^2}
\]  

(5.1)

\( \Delta p \) is the difference of average static pressure at inlet and the outlet pass while \( U_{in} \) is the area averaged velocity magnitude at the inlet of the geometric domain shown in Fig 4-1. Figure 5-1 compares the relative pressure drop of the channel, as defined by Eq. 5.1, for an aspect ratio of inlet pass \( (W_{in}/H) \) equal to 1:3 with a channel having aspect ratio of 1:2. The comparison is made at different divider-to-tip wall distances \( (W_{el}) \) normalized with channel inlet width \( (W_{in}) \) as well as with its height \( (H) \). Normalizing with \( W_{in} \) shows the influence of \( W_{el} \) on pressure drop. Thus it can be seen that for both cases the pressure drop decreases non-linearly with increasing divider-to-tip wall distance. While normalizing \( W_{el} \) with \( H \) shows the influence of \( W_{in} \) (or in other word inlet channel aspect ratio) on the pressure drop which shows that with decrease in aspect ratio, the pressure drop decreases. But as the divider-to-tip wall distance increases, this difference in pressure drop for two aspect ratio channels decreases. From \( W_{el}/H = 0.5 \) to 1, this difference in \( \Delta p^* \) decreases from 0.7 to 0.2. For low aspect ratio case, it appears to attain a constant value after the ratio \( W_{el}/H \) reaches 0.88 or \( W_{el}/W_{in} \) reaches 2.63.
Figure 5-1: Effect of $W_{el}$ normalized with the height of the channel $H$ (on the left) while normalized with inlet pass width $W_{in}$ (on the right) on the relative pressure drop for two-pass smooth channel.

Contours of $\text{Nu}_{\text{avg}}/\text{Nu}_{\text{st, avg}}$ for different cases of $W_{el}/W_{in}$ are shown in Fig. 5-2. For the heat transfer analysis, the channel was split into three sections as shown in Fig. 5-2: inlet pass (faces 1a, and 2), bend (faces 1b, 3, 6b and 4) and outlet pass (faces 5 and 6a). This was done to facilitate comparison with Shevchuk et al. [2011], where the averaged Nusselt number was defined using the volume averaged temperature of each section. Increase in the divider-to-tip wall distance shifts, the region of enhanced heat transfer from side walls (faces 6b and 6a) and outlet pass (6), towards the section of bend region attached to the inlet pass. This accompanies a minor increase in the values of $\text{Nu}_{\text{avg}}/\text{Nu}_{\text{st, avg}}$ in the inlet. This means that with an increase in $W_{el}/W_{in}$ ratio, the flow impingement onto the outlet pass bottom (6) and external side wall (6a and 6b) weakens, while the vortices get strengthened in the bend.
In order to study the effect of decrease in aspect ratio of the inlet pass, the quantitative results for the averaged Nusselt number at different divider-to-tip wall distances are compared with the numerical results by Shevchuk et al. [2011] and are shown in Fig. 5-3. Again the divider-to-tip wall distance has been normalized with the channel inlet width \( W_{\text{in}} \) showing the influence of divider-to-tip wall distances \( W_{\text{d}} \) as well as channel height \( H \) which shows the effect of change in aspect ratio. The first observation is that, with a decrease in aspect ratio from 1:2 to 1:3, the heat transfer at the bend as well as at the outlet pass decreases. For a channel with the inlet pass aspect ratio equal to \( W_{\text{in}} / H = 1:3 \), the heat transfer at the outlet pass decreases with increase in divider-to-tip wall distance. Whereas in the bend, the heat transfer first decreases with an increasing gap but then increases and settles at a constant value. The comparison shows that for a channel with \( W_{\text{in}} / H = 1:2 \), heat transfer at bend increases and then decreases with increase in \( W_{\text{d}} / H \). In contrast, for a channel with \( W_{\text{in}} / H = 1:3 \), heat transfer decreases initially and then increases to attain a constant value independent of \( W_{\text{d}} / H \).
To understand the reason for the differences in the heat transfer results for different inlet channel aspect ratios, velocity magnitudes at the symmetry and near the wall are compared for a case where $W_e/W_{in}$ is equal to 1.5. The velocity magnitudes for both aspect ratio channels are shown in Fig. 5-4. For both cases, the velocity magnitudes at symmetry are different then close to the bottom wall. This shows the three dimensional nature of the flow. For the channel with a low aspect ratio at the inlet, the velocity distribution near the bottom wall is non uniform. As the divider-to-tip wall distance is small for the low aspect ratio channel ($W_e/W_{in}$ is constant for both cases), the fluid seems to accelerate more in the bend region. This acceleration is more prominent at the symmetry plane. Due to the three dimensional nature of flow, this acceleration dies out near the wall. In contrast to that, for the high inlet aspect ratio channel, the flow becomes more uniform near the wall resulting in higher heat transfer in the bend region. At the outlet pass, for the low inlet aspect ratio channel, the flow accelerates more than that for the high inlet aspect ratio channel. This is the reason for high heat transfer at the outlet pass.
Figure 5-3 and 5-4 also show that for same $W_{\text{el}}/W_{\text{in}}$ ratio, the channel with low inlet aspect ratio has higher heat transfer at the outlet-pass compared to the one with high inlet aspect ratio. The trend is opposite at the bend region.

5.1.2 Overall Comparison

The divider-to-tip wall distance, $W_{\text{el}}$, affects not only the pressure drop in the channel but also the heat transfer at the bottom surface of the bend as well as at the tip of the channel. To evaluate the combined effect of heat transfer enhancement and the associated pressure drop, a parameter called thermal performance is calculated as shown in Eq. 5.2.

$$\eta = \frac{N_{\text{u,avg}} / N_{\text{u,o}}}{(f/f_o)^{1/3}}$$  (5.2)

This parameter is helpful to express how much a pressure loss is spent to reach a certain level of heat transfer. A high value of this parameter means that more heat transfer enhancement at low pressure drop is achieved. This is used to compare the heat transfer enhancement performance at the bend region (face 3 and 4). The results for thermal performance of the bend region are presented in Fig. 5-5. This trend is a combined result from the trends for $\Delta p^*$ (Fig. 5-1) and $N_{\text{u,avg}}/N_{\text{u,o}}$. The pressure drop attains a constant value at $W_{\text{el}}/H = 0.88$, as shown in Fig. 5-1. The thermal performance at bend region continues to increase until $W_{\text{el}}/H = 0.88$, after which it tends to attain a constant value. This is the value of the divider-to-tip wall distance, for a channel with the inlet pass aspect ratio $W_{\text{in}}/H = 1:3$, where maximum heat is transferred from the bend region at a minimum pressure drop.
Figure 5-5: Thermal performance at bend (faces 3 and 4) of the smooth two-pass channel with aspect ratio of $W_{in}/H = 1:3$ and $W_{out}/H = 1:1$ for different $W_{el}/H$.

5.2 Two-Pass Ribbed Rectangular Channel

5.2.1 Pressure Drop and Heat Transfer

Similar to the case of smooth channel, the relative pressure drop $\Delta p^*$ has been calculated for the ribbed channel using Eq. 5.1. $\Delta p$ is the difference of average static pressure at inlet and a plane located just at the end of the fourth rib in the outlet pass. $U_{in}$ is the area averaged velocity magnitude at the inlet of the geometric domain shown in Fig 4-6. It was done to observe the effect of varying the divider-to-tip wall distance ($W_{el}$) on the pressure drop. Fig. 5-6 compares the relative pressure drop of the channel, where the aspect ratio of the inlet pass ($W_{in}/H$) is equal to 1:3 with the channel having an aspect ratio equal to 1:2 at the inlet by Shevchuk et al. [2011]. Both studies used the same turbulence model, same near wall treatment, comparable grid sizes, same temperature boundary conditions on the wall, inlet profiles of velocities, turbulences and temperature computed using a periodic segment placed upstream of the two-pass channel, same data reduction and post-processing. It shows that for both cases, pressure drops initially with increase in $W_{el}$. But the influence of divider-to-tip wall distance on pressure drop minimizes with further increase. Also decrease in aspect ratio of the inlet pass has significant effect on reduction of pressure drop.
The channels have been divided into different regions which were numbered accordingly. Figure 5-7 shows one of the channels, where the divider-to-tip wall distance ($W_{el}$) is equal to 131 mm. The Nusselt number predicted by the Dittus-Boelter correlation has been used as a reference value to make a comparison.

Figure 5-8 presents the averaged Nusselt numbers normalized by the Nusselt number obtained by Dittus-Boelter correlation ($\text{Nu}_o$) for the bend (face 3), the area between the first and the second ribs (face 5), the area between the second and the third rib (face 6) and the area between the third and the fourth rib (face 7).
The periodicity in the velocity and temperature fields in obviously expected to be established in the outlet pass, if the model included more ribs (minimum five). However a sign of this trend is observed after the second rib even with this simplified model. It can be observed that for smaller $W_e/H$ ratios the periodicity is approached after the first rib, but with the increase in this ratio, the periodicity moves downstream and can be noticed after the second rib instead. This information is very useful, as it can reduce the computational domain in multi-pass channels. Only certain restricted numbers of ribs in the outlet pass would be sufficient to simulate, and the results from output of one outlet pass can be mapped as inlet conditions to the following inlet pass. It should also be noted that such a periodicity was not reported by Shevchuk et al. [2011]. This may mean that in order to get periodic conditions in the outlet pass on a two-pass ribbed channel, the number of ribs in the computational domain plays a critical role.

Figure 5-9 shows the Nusselt number distribution for faces 3 and 4. It is predicted that for a channel with small aspect ratio inlet pass, heat transfer at bend bottom (face 3) is less, although for both cases it decreases with increase in divider-to-tip wall distance. For face 4, which is the tip of the channel, a smaller aspect ratio channel has lower heat transfer value at small divider-to-tip wall distance but the decay in heat transfer (with increase in divider-to-tip wall distance) is exponential such that at higher $W_e/H$ the heat transfer is more than that of a channel with high aspect ratio inlet pass. Also it is worth to note that in contrast to large aspect ratio channel, the Nusselt number at face 4 never approaches that of face 3 in the given range of $W_e/H$ for small aspect ratio channel.
Figure 5-9: Effect of $W_{el}$ on average heat transfer on the bend bottom (face 3) and tip wall (face 4 in the ribbed channel, different varying $W_{in}/H$.

Figure 5-10 shows the Nusselt number distribution for outlet bottom (face 5+6+7) and side wall (face 8) with varying the divider-to-tip wall distance for the case where the aspect ratio of the inlet pass is $W_{in}/H = 1:3$. Note that while averaging the Nusselt numbers for outlet bottom, the rib area and heat transfer was not included. It is predicted that for such a channel, the heat transfer at the outlet bottom increases initially, with increase in $W_{el}/H$ but then it remains constant with standard deviation of about 5%. For the side wall, the heat transfer decreases with increase in $W_{el}/H$.

Figure 5-10: Effect of the $W_{el}$ on average heat transfer on different faces in the ribbed channel, varying $W_{in}/H = 1:3$ to $W_{out}/H = 1:1$. 
To explain the heat transfer behaviour at different portions of the channel, contours of velocity magnitudes near the wall (one node distance) and at the symmetry plane for the smallest and largest $W_{in}/H$ cases have been plotted and shown in Fig. 5-11. The first node is in the boundary layer so the velocities are very small near the wall compared to the free stream velocity at the symmetry plane. This forces to keep different scales of velocity magnitudes at two different locations. Despite that, it is very clear that the flow is three dimensional. The contours near the wall shows that the small divider to tip wall distance ($W_{el}/H$) results in higher velocities compared to larger divider to tip wall distance at the same flow rates at bend while the contours at the symmetry plane shows that the fluid impinges on the tip and side wall with higher velocity in case of small divider-to-tip wall distance. The strong bend induced secondary flow (Dean Vortices) due to high velocities in small $W_{el}/H$ case results in higher heat transfer at the bend bottom while the high velocity impingement at the tip and side wall thus results in high heat transfer at these regions for smaller $W_{el}/H$.

Figure 5-11: Contours of Velocity magnitude near the bottom wall and at symmetry plane of two different divider-to-tip wall distances for channel with $W_{in}/H = 1:3$.

5.2.2 Overall Comparison

As calculated for smooth channel, thermal performance of the ribbed two-pass channel with $W_{in}/H = 1:3$ and $W_{out}/H = 1:1$ has been calculated using Eq. 5.2. The thermal performance of bend bottom (Face 3), tip (Face 4), outlet pass bottom (Face 5+6+7) and side wall (Face 8) is shown in Fig. 5-12. The trends represent the combined effect of heat transfer from the individual
wall and the pressure drop across the channel. Due to difference in the heat transfer from each wall, the thermal performance is also different. With varying divider-to-tip wall distance, the trends are similar for each wall. It can be concluded that although the heat transfer increases with the decrease in divider-to-tip wall distance for bend region as well as the side wall, $W_{el}/H = 0.5$ exhibits the lowest thermal performance due to the high pressure drop as shown in Fig. 5-6. The optimum value of divider-to-tip wall distance is found to be $W_{el}/H = 0.63$ for this channel. In contrast to that Shevchuk et al. [2011] has reported that for two-pass rectangular channel with $W_{in}/H = 1:2$ and $W_{out}/H = 1:1$, the optimum value for divider-to-tip wall distance with reference to bend bottom and tip is $W_{el}/H = 0.5$.

\[Figure 5-12: \text{Thermal performance at different divider-to-tip wall distances for channel with } W_{in}/H = 1:3.\]
6. RESULTS FROM TRAPEZOIDAL CHANNEL SIMULATIONS

In this section the results for heat transfer and pressure drop in different two-pass trapezoidal channels are presented. The aim is to figure out the best channel design in terms of heat transfer enhancement along with the lowest possible pressure drop. The details of the numerical and physical models can be found in chapter 4.

Figure 6-1 shows the Nusselt number distribution normalized with the Nusselt number obtained by the Dittus-Boelter correlation (Eq. 4.1) on the bottom wall, tip wall and trailing edge of the smooth channel. In a smooth channel, fluid impinges on the tip wall, which results in a high heat transfer region. The turn at the bend creates secondary flow that causes a high heat transfer spot at the bottom wall in the bend region. A high heat transfer region at the trailing edge wall attached to the bend region is the result of impingement onto the trailing edge wall. Similarly, a region of high heat transfer is observed at the bottom wall upstream of the outlet pass. Further downstream, the effect of bend on flow reduced, and more uniformly distributed heat transfer regions are observed at the bottom wall and trailing edge.

Figure 6-1: Contours of Nu/Nuo for the two-pass smooth trapezoidal channel.

Figure 6-2 shows the Nusselt number distribution normalized with the Nusselt number obtained by the Dittus-Boelter correlation (Eq. 4.1) on the bottom wall, tip wall and trailing edge of the ribbed two-pass trapezoidal channel. For this case, the impinging effect is enhanced at the tip wall part close to the inlet pass. The bottom wall in the bend region is supplied with a rib. Upstream of this rib, heat transfer is enhanced at the bend bottom near the tip wall, and reduced over the rest of this region downstream of the inlet pass. This reduction in heat transfer is caused by the presence of ribs in the inlet pass. At the bend bottom, the presence of a rib results in the ribbed induced secondary flow that interacts with the bend induced secondary flow (Dean Vortices). This interaction appears to be favourable for enhancing heat transfer in the bend. The presence of ribs on the bottom wall of the outlet pass not only enhances heat transfer on the
bottom wall itself, but also on the trailing edge wall. In the outlet pass downstream of the bend region, the rib induced secondary flows starts to counteract with the Dean vortices and reduce the impingement effect on the bottom wall. Also, low heat transfer regions start to appear in the outlet pass upstream of the ribs because of the local flow deceleration.

**Figure 6-2:** Contours of $\text{Nu}/\text{Nu}_0$ for the two-pass ribbed trapezoidal channel.

Figure 6-3 shows the Nusselt number distribution normalized with the Nusselt number obtained by the Dittus-Boelter correlation (Eq. 4.1) on the bottom wall, tip wall and trailing edge of the channel with ribbed trailing edge only. In the bend and the inlet pass, the local heat transfer distribution depicted in Fig. 6-3 is practically the same as that for a channel with the ribbed bottom wall given in Fig. 6-2. Qualitatively the effect of ribs on the trailing edge heat transfer in Fig. 6-3 looks similar to the effect of ribs on the bottom wall heat transfer in Fig. 6-2.

**Figure 6-3:** Contours of $\text{Nu}/\text{Nu}_0$ for the two-pass trapezoidal channel with ribs on the trailing edge only (Case A).
Figure 6-4 shows the Nusselt number distribution normalized with the Nusselt number obtained by the Dittus-Boelter correlation (Eq. 4.1) on the bottom wall, tip wall and trailing edge of the channel with ribbed bottom wall and trailing edge (inline). The inline arrangement of ribs on the bottom and the trailing edge (Case B) results in the cumulative effect in the bend and after the first rib in the outlet pass. After the second rib, heat transfer drastically reduces because ribs destroy bend induced Dean Vortices, which would otherwise favour heat transfer enhancement.

Figure 6-5 shows the Nusselt number distribution normalized with the Nusselt number obtained by the Dittus-Boelter correlation (Eq. 4.1) on the bottom wall, tip wall and trailing edge of the channel with ribbed bottom wall and trailing edge (staggered). The comparison between Fig. 6-4 and 6-5 shows that there is more enhancement on the bottom wall in case of staggered arrangement but on the trailing edge this arrangement results in low heat transfer regions after the second rib.

**Figure 6-4:** Contours of $\frac{Nu}{Nu_0}$ for the two-pass trapezoidal channel with ribs on the bottom wall as well as on the trailing edge with inline arrangement (Case B).

**Figure 6-5:** Contours of $\frac{Nu}{Nu_0}$ for the two-pass trapezoidal channel with ribs on the bottom wall as well as on the trailing edge with staggered arrangement (Case C).
To provide an overall measure of enhancement, area averaged Nusselt numbers were calculated for all walls in the outlet pass. Figure 6-6 presents these results, where the area averaging has been taken for the walls including ribs as well as without ribs, which is helpful in separating the enhancement due to increase in the heat transfer area and that due to the ribbed induced secondary flow. The results show that for the outlet pass there is total enhancement of 24% in the Nusselt number for Case C compared to that of a smooth channel, if ribs are taken into consideration. If the rib area is excluded, this enhancement is about 12%. This shows the share of enhancement due to ribbed induced secondary flow. Further study on the placement of the ribs and rib’s geometry can lead to more enhancement of heat transfer. But these results show that the trailing edge can be cooled with internal convection only. This can than lead to design a channel where steam is used as coolant.

The ribs act as an obstacle to the flow and increase the pressure losses. Equation 4.3 was used to calculate the friction factor that represents the pressure drop across the channels. Equation 4.4 was used to normalize the results shown in Fig. 6-7. Due to more obstacles, Case B results in about 36% higher pressure drop compared to a smooth channel. The staggered arrangement reduces this pressure drop by about 4%.
The presence of ribs may enhance heat transfer, but also involves increased pressure losses. To express how much pressure losses are associated with a certain level of heat transfer, a parameter called thermal performance is calculated defined by Eq. 5.2.

Figure 6-8 presents the thermal performance for the outlet pass (including and excluding the rib’s area in averaging the Nusselt number). The staggered arrangement of ribs shows best results for the outlet pass with an increase of 7% in thermal performance, if the rib area is considered. However, thermal performance reduces when ribs are excluded in calculating the area-averaged Nusselt numbers. This shows that the ribs are causing more obstacle than enhancing the heat transfer. Note that this observation is about the average Nusselt numbers. While analyzing the local distribution, shown in Fig. 6-1 to 6-5, it was observed that after rib 1 and 2, there is rib induced secondary flow resulting in the enhancement of heat transfer but it weakens later on, which in total causes the ribs to be ineffective in enhancing heat transfer.
The objective of the new designs of trapezoidal channels was to enhance the heat transfer in the trailing edge since that is a critical life limiting region especially when trailing edge slots are eliminated. Table 6.1 presents the area-averaged Nusselt number normalized with the Nusselt number by the Dittus-Boelter correlation and thermal performance for the trailing edge for all cases. This also shows that ribs are not helpful in enhancing heat transfer at the trailing edge, and the smooth wall not only has comparable heat transfer values with other cases but also has the least pressure drop. This results in the highest thermal performance for the smooth channel. The results also show that ribs at the trailing edge are enhancing heat transfer due to an increase in heat transfer area rather than the ribbed induced secondary flow.

**Table 6-1:** Area averaged Nu/Nu₀ and thermal performance for trailing edge of the trapezoidal channels.

<table>
<thead>
<tr>
<th></th>
<th>Nu/Nu₀</th>
<th>η</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>With ribs</td>
<td>Without ribs</td>
</tr>
<tr>
<td>Smooth</td>
<td>2.63</td>
<td>1.19</td>
</tr>
<tr>
<td>Ribbed</td>
<td>2.69</td>
<td>1.12</td>
</tr>
<tr>
<td>Case A</td>
<td>2.72</td>
<td>1.16</td>
</tr>
<tr>
<td>Case B</td>
<td>2.72</td>
<td>1.06</td>
</tr>
<tr>
<td>Case C</td>
<td>2.49</td>
<td>0.98</td>
</tr>
</tbody>
</table>
The thermal performance for the bottom wall for a smooth and a ribbed bottom wall channel for the region shown in Fig 6-1 and 6-2 according to the experimental data by Lee et al. [2007] is shown in Table 6-5 along with the numerical results.

**Table 6-2: Thermal performance for the bottom wall of a smooth and a ribbed bottom wall channel.**

<table>
<thead>
<tr>
<th></th>
<th>Experiment</th>
<th>Numerical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Smooth</td>
<td>1.0</td>
<td>0.8</td>
</tr>
<tr>
<td>Ribbed</td>
<td>1.1</td>
<td>0.9</td>
</tr>
</tbody>
</table>

The experimental thermal performances are found to be better at the bottom wall for the ribbed channel than that of the smooth channel. The numerical results suggest the same though the numerical value is different. This is opposite to what was observed for the trailing edge. It was observed during validations that the Nusselt numbers were underpredicted in the ribbed channel by about 17% whilst for the smooth channel it was overpredicted by about 7% in average. Therefore it was concluded that the thermal performance is also underpredicted in case of the ribbed channels.

Although the numerical model underpredicts the Nusselt numbers in the ribbed channel, which results in the prediction of better thermal performance of the smooth channel, it is worthwhile to compare the three new designs for the enhancement of the trailing edge heat transfer. As shown in Table 6-1, the heat transfer enhancement is similar in Cases A and B which is higher than that of Case C. But Case B represents higher pressure losses leading to low thermal performance than Case A. Therefore it is concluded that, for the trailing edge perspective, Case A seems to be the better option. But as shown in Fig. 6-1, for overall perspective of the outlet pass, Case C is much better than Cases A and B, if pressure losses are acceptable. Therefore, use of internal convection only is possible for cooling of trailing edge of the blade if staggered arrangement of ribs at the bottom wall and trailing edge wall is used.
7. RESULTS FROM CONJUGATE HEAT TRANSFER SIMULATIONS

Air extracted from the last stages of the compressor has been used as a cooling medium in gas turbines for years now. The extraction of air from the compressor results in a reduction in overall performance of the turbine. In combined cycle power plants, steam is available which can be used as the cooling medium replacing air. Steam has better thermal properties compared to air which makes it an attractive choice as a coolant. To show its supremacy on air, a comparison of the performance of both coolants has been made, where steam as well as air has been used as coolant in a two-pass smooth channel shown in Fig. 4-16 under similar thermal conditions.

Five different coolant flow rates were selected such that the Reynolds number at the inlet of the channel ranges from 20,000 to 100,000. The temperature and pressure at the inlet was the same for both air and steam resulting in the Prandtl number of steam equal to 0.94, while for air it was 0.68. As the thermal properties of air and steam are different at the same pressure and temperature, the velocity (and mass flow rate) at the inlet is different at the same Reynolds number such that under the selected conditions the velocity of air was found higher than that of steam at the inlet. Normalized temperature profiles at trailing edge corner are shown in Fig. 7-1 for all the cases.

![Normalized temperature profile at trailing edge corner for two-pass smooth channel at different Reynolds number for air and steam.](image-url)
One clear observation is that with the increase in flow rate, heat transfer increases and thus the temperature values drop for either of two coolants. Although the velocity at inlet was less when steam was used, the drop in temperature magnitude is more compared to air at the same Reynolds number. It is also worth to note that the temperature profile one gets at higher Reynolds number using air, is reproduced using steam at low Reynolds number. This shows that at similar thermal conditions, the required flow rate of steam is lower than that of air to achieve the same cooling. As an example, the temperature profile achieved by using air at Reynolds number = 80,000 was reproduced by using steam at Reynolds number = 60,000. Thus there is a reduction of 25% in required Reynolds number. This corresponds to a reduction of 45% in mass flow rate under given conditions.

7.1 Heat Transfer

Nine different two-pass trapezoidal channels are compared under the same thermal conditions. The purpose is to select the design that yields the lowest metal temperatures. The hottest region of the channel is the corner of the trailing edge called trailing edge corner in Fig. 4-16. The normalized temperatures on that corner are plotted for all of the nine cases and shown in Fig. 7-2. Point 0 is the edge point, where tip wall and the trailing edge meet. As for all cases, the inlet pass and the turn region are the same, the normalized temperature values are almost the same for all the cases at side wall attached to the turn region (Z/Z_max = 0 to 0.05). The effect of different heat transfer enhancement methods is more visible after this length. All ribbed channels except Case C, results in a notable reduction in the wall temperature. The temperatures are not only reduced but also approach a constant value along the length which is desired in reducing the thermal gradients in the metal. Surprisingly, neither of the finned channels with smooth bottom walls (Case F, G and H) leads to a reduction in metal temperatures. In fact, it can be observed that the bigger the fin is, the more ineffective it is at reducing the metal temperature. Another notable observation is that the channel with ribs at the trailing edge only (Case C) and smooth channel have similar results. This indicates that the ribs used at the trailing edge are not helping much in increasing heat transfer. On other hand, the ribs at the bottom wall are more helpful in reducing metal temperatures (or in other words increasing the heat transfer). Interestingly, providing ribs or a fin at the trailing edge has no significant effect, as these produce similar results as the channel with the ribs at bottom wall only.

For all channels, the normalized temperatures are more than the target value of T/T_{targ} = 1 along the length. This means that none of the channel design is acceptable. Therefore further changes in the design are required to achieve the goal.
7.2 Pressure Drop

Turbulators and other heat transfer enhancement methods may result in an increase in heat transfer and a reduction of the metal temperatures but they can increase the pressure drop. Thus it is very important to compare the pressure drop for these designs. Static pressure values were monitored at two points. Point 1 is at a distance of 14 mm from the tip wall to the inlet pass while point 2 is at a distance of 70 mm from the tip wall to the outlet pass. Equation 4.3 is used to calculate the friction factor $f$, where $l$ is the length at the symmetry plane from one point of pressure measurement to the other point. $A_c$ and $D_h$ are the cross sectional area and hydraulic diameter respectively at station 1. The friction factor was normalized with $f_o$, the reference friction factor for fully developed turbulent flow through a smooth channel, given by Eq. 4.4. Figure 7-3 presents the results for the friction factor ratio calculated in this way for all cases.

Figure 7-2: Temperature profiles at trailing edge corner for all cases.
The lowest pressure drop is observed for Case F (smooth channel with small fin) but the heat transfer enhancement for this case is not significant. It was observed from Fig. 7-2 that the channels with ribs at the bottom wall result in a reduction in metal temperature. So comparing the pressure drop associated with those reveals that Case E (ribs in staggered arrangement) offers a small drop in pressure in comparison. Thus it is concluded, that for higher heat transfers with lower pressure drops, a channel with ribs at the bottom wall as well as at the trailing edge in staggered arrangement is suitable for further improvements.

7.3 Variation in Divider Wall

It is observed that for all cases, the normalized temperature values are above the target value of 1, not only at the corner but also along the length of the trailing edge corner. Therefore more innovative designs are required to achieve the goal. For internal turbulent flows Nusselt number is defined as

$$Nu = f(Re, Pr)$$  \hspace{1cm} (7.3)

where

$$Nu = \frac{hD_h}{k}$$  \hspace{1cm} (7.4)

$$Re = \frac{\rho UD_h}{\mu}$$  \hspace{1cm} (7.5)
These relations suggest that for constant physical properties, heat transfer increases with an increase in fluid velocity. If the flow in the outlet pass is accelerated somehow, wall temperatures can be reduced. Two variations in the divider wall orientation were tested that aim to accelerate the flow in the outlet pass to reduce wall temperatures in this region where they are highest. On the left hand side in Figure 7-4, a two-pass smooth channel is shown which has a tapered divider wall while on right hand side; a two-pass smooth channel with a tilted divider wall is shown. In the tapered case, the divider wall is altered such that the width of the inlet pass remains the same and is constant along the length as in the base case (Case A), but the outlet pass becomes a convergent channel. The maximum width of the divider wall is 4 mm, while it is 0.8 mm wide on the other end. In the other case, the divider wall is tilted with an angle of 88° thus making the inlet as well as the outlet pass as converging channels. The maximum width at inlet is 5.86 mm while that at outlet the maximum width is 7.91 mm.

The velocity magnitudes at the symmetry plane of these two cases are shown in Fig. 7-5. The Reynolds number, at the inlet is same for both channels therefore at the inlet of the tilted divider wall channel which has large hydraulic diameter; the velocities are low compared to those at the inlet of the tapered divider wall channel with smaller hydraulic diameter. The converging channels results in increase in velocities, as expected. For the channel with tilted divider wall, the flow is more accelerated in the outlet pass as it follows an accelerated flow from the converging inlet pass.
The effect of this accelerated flow can be seen in Fig. 7-6, which presents the normalized temperature profile at trailing edge corner for these two cases and compares it with that of the smooth channel (Case A). The flow conditions at the turn region of a smooth channel (Case A) and channel with tapered divider wall are similar therefore the temperature profiles for the two are also similar initially. The accelerated flow in the outlet pass results in a drop in temperature. In case of the channel with tilted divider wall, the flow at the turn is more accelerated therefore it results in a higher temperature drop.

Although the smooth channel with tilted divider wall helps in reducing the temperature values significantly, still the values are above the target value of 1. Therefore more improvements are needed in the design. The best option is to introduce a staggered arrangement of ribs at the outlet.
pass of this tilted divider wall channel. The reason is that a staggered arrangement of ribs provides the least pressure drop. Figure 7-7 presents the velocity magnitudes at the symmetry plane of such a combination. The accelerated flow along with the ribbed induced secondary flow is expected to improve heat transfer.

![Figure 7-7: Velocity magnitudes at the symmetry plane of two-pass channel with staggered arrangement of ribs and tilted divider wall.](image)

The result of this combination is shown in Fig. 7-8, which presents the normalized temperature profile at trailing edge corner of this channel and compares it with staggered ribbed channel (Case E). The new arrangement is helpful in reducing the wall temperature below the target value except at the initial length of $Z/Z_{\text{max}} = 0$ to 0.08.

![Figure 7-8: Temperature profile at trailing edge corner for a staggered ribbed channel and its variation of divider wall.](image)
It is vital to see if other regions of the channel are also in an acceptable range or not. Figure 7-9 shows the contours of the normalized temperature at different walls of the channel. On the left hand side, the scale is the same as in the previous figures. To get a clear picture, the scale is narrowed and is shown on the right hand side. It is observed that except for a small region in the corner (where the bottom wall, trailing edge and tip meet) the temperature values are in acceptable range.

Figure 7-9: Normalized temperature contours at different walls of the channel with tilted divider wall and staggered arrangement of ribs.

7.4 Impingement at Corner

Figure 7-8 and 7-9 show that a channel with tilted divider wall and staggered arrangement of ribs in the outlet pass helps in reducing the temperature values significantly to the target value in most of the regions of the trailing edge. But still the corner of the channel is not in the acceptable temperature range. Therefore a more aggressive method is needed to cool the corner. One such method is to impinge the flow on that surface. Figure 7-10 shows the top view of three different configurations which aim to impinge the flow on the corner, keeping the tilted divider wall since it has proved to be effective in reducing the wall temperatures. The purpose is to see the effect of the impingement on the corner only; therefore, these variations are made in the smooth channel to reduce the complexity of the problem. Case 1, is a smooth channel with a tilted L-shaped divider wall where the angle between the two legs is 60°. This L-shape helps direct the fluid to the corner. The maximum distance L₁, of this divider wall from the tip is 5.22 mm while the minimum distance L₂, is 4.44 mm. The L-shaped divider wall is truncated at the end such that its distance from the trailing edge I₁, is 1.5 mm while I₂, is 2 mm. The purpose of this truncation is to allow the flow to diffuse evenly in the outlet pass. Case 2 and Case 3 have the same dimensions except that different types of openings in the L-shaped divider wall are introduced. The reason is to eliminate the low pressure zone expected in Case 1. The opening in Case 2 is tapered such that the maximum width of the opening is 1.75 mm while the minimum width is 0.77 mm. For Case 3, an opening with constant width of 0.7 mm is provided. The end of the divider wall before the opening is wedge shaped.
The velocity contours at the symmetry plane of these three cases is shown in Figure 7-11. For Case 1, the fluid impinges at the corner with high velocity. It then forms a wall jet and continues to flow along the trailing edge in the outlet pass. The high velocity region in the outlet pass which is attached to the divider wall is in the reverse direction. This results in a large recirculation zone in rest of the outlet pass, hence a high pressure drop is expected in this design. The opening present in the divider wall, as in Case 2 and Case 3, helps in eliminating this big wake. But by doing this, the mass flow is divided and the velocity at the corner is not that high (compared to Case 1) which reduces the impinging effect at the corner. For Case 2, the flow through the opening mixes more evenly with the flow at the outlet pass compared to Case 3. This results in more evenly distributed velocity contours in the outlet pass in Case 2.
The effect of impingement on heat transfer at the corner of the channel is shown in Fig. 7-12 which presents the normalized temperature profiles at trailing edge corner for the three impingement cases. The high impinging velocity at the corner in Case 1 results in a significant temperature drop. In fact not only the temperature values at the corner are below the target value but also these are below it all along the length of the channel. So this design fulfils the design target even without the use of any heat transfer augmentation devices like ribs etc. Case 2 and 3 are also providing sufficient impingement and the normalized temperature values are below the target value for most of the corner region as well as most of the channel length.

Figure 7-12: Normalized temperature profile at trailing edge corner for three cases of impingement cooling of smooth channel.

Figure 7-13 presents the normalized temperature contours at different walls for the cases defined in Fig. 7-10. As for trailing edge corner, Case 1 is capable of producing sufficient heat transfer which reduces the wall temperature to below the target value at every part. For Case 2 and 3, a small part of the corner and a small region of the trailing edge and bottom wall near the outlet are above the target value.
From the above results, Case 1 is the only channel design which fully satisfies the design target as far as heat transfer is considered. But as discussed earlier, this design has a large recirculation region in the outlet pass and that can cause a large pressure drop. Therefore, pressure drop as defined by Eq. 7.1 and 7.2 are calculated for the three channels. The normalized friction factor for these cases is given in Table 4.1.

**Table 7-1: Normalized friction factor for three cases of impingement cooling of smooth channel.**

<table>
<thead>
<tr>
<th></th>
<th>$(f/f_0)^{1/3}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impingement_Case 1</td>
<td>30.04</td>
</tr>
<tr>
<td>Impingement_Case 2</td>
<td>11.48</td>
</tr>
<tr>
<td>Impingement_Case 3</td>
<td>13.67</td>
</tr>
</tbody>
</table>

As expected, the pressure drop in Case 1 is much higher than the other two cases. Though the same channel is very effective in reducing the wall temperatures to an acceptable range, it is unacceptable due to its high drop in pressure. Therefore, Case 2 which offers the least drop in pressure should be altered such that the wall temperature at the trailing edge reduces to the acceptable range. A channel with staggered arrangement of ribs and a tilted divider wall fulfil this requirement as shown in Fig. 7-8 and 7-9. Therefore a fusion of the two should be an effective way to achieve the target with the least pressure drop. Figure 7-14 shows the velocity magnitude at the symmetry plane of a channel having L-shaped divider wall with a tapered opening and staggered ribs in the outlet pass. The fluid impinges at the corner and mixes with the fluid coming from the opening in the divider wall. The ribs present in the outlet pass produce ribbed induced secondary flow and results in the mixing of the fluid.
Figure 7-14: Velocity magnitude at the symmetry plane of a channel having L-shaped divider wall with a tapered opening and staggered ribs in the outlet pass.

The combined effect of the ribs in the outlet pass and the impingement in the corner along with the tilted divider wall results in the required temperature drop. Figure 7-15 compares the normalized temperature profile at trailing edge corner for the case with staggered arrangement of ribs and straight divider wall, the case with staggered arrangement of ribs and tilted divider wall and the case with staggered arrangement of ribs and impingement in the corner. A tilted divider wall with impinging effect with staggered arrangement of ribs and a tapered opening in the divider wall is the final arrangement for the design target set for this study. Only a small region of length $Z/Z_{\text{max}} = 0$ to $0.02$ is above the target value.

Figure 7-15: Normalized temperature profile at trailing edge corner for three different types of channels with staggered arrangement of ribs.

Figure 7-16 presents the normalized temperature contours for different walls of the final designed channel. A narrower scale is used on the right hand side. A very small area at the corner
is above the target value. Thus the design target is safely achieved in most of the region by using this kind of channel.

![Figure 7-16: Contours of normalized temperature at different walls of the channel with staggered arrangement of ribs and impingement in the corner.](image)

### 7.5 Thermal Performance

The different heat transfer techniques used, generally result in an increase in pressure drop. An optimized cooling channel is one which results in maximum heat transfer with minimum pressure drop. Thermal performance of different configurations of the channels is calculated using Eq. 5.2. The area averaged Nusselt numbers are calculated for all inner walls of the channel (which are in contact with the fluid). Table 7-2 shows the thermal performance and pressure drop across the different channels considered in this study.

| Table 7-2: Pressure drop and thermal performance of different two-pass trapezoidal channels. |
|-----------------------------------------------|-----------------|-------------------|
| Smooth channel (Case A)                       | 2.33            | 0.56              |
| Staggered ribbed channel (Case E)             | 2.63            | 0.69              |
| Smooth channel with tilted divider wall        | 6.81            | 0.39              |
| Staggered ribbed channel with tilted divider wall | 9.06      | 0.28              |
| Smooth channel with Impingement (Case 1)      | 30.04           | 0.14              |
| Smooth channel with Impingement (Case 2)      | 11.48           | 0.26              |
| Smooth channel with Impingement (Case 3)      | 13.67           | 0.25              |
| Staggered ribbed channel with impingement     | 12.79           | 0.20              |

Although the design target is achieved by a channel with a tilted but L-shaped divider wall, which has a converging opening in it and a staggered arrangement of ribs in the outlet pass (staggered ribbed channel with impingement) such an arrangement results in very low thermal
performance which is due to the high pressure drop across the channel. The smooth channel with L-shaped divider wall (Impingement Case 1) is capable of achieving the design target even without using any ribs but results in very high pressure drops. Thus the thermal performance of this channel is the worst amongst all cases. Figure 7-8 and 7-9 show that the channel with tilted divider wall and staggered arrangement of ribs also results in wall temperatures that fall below the design target value of $T/T_{\text{targ}} = 1$ over most of the channel. It has similar temperature profile at trailing edge corner as the staggered ribbed channel with impingement, shown in Fig. 7-15, except that this case has a larger area at the corner which is above the target value. But the thermal performance of this channel is better as it offers a lower pressure drop. Thus with use of thermal barrier coating or truncation of this area, the channel can be altered to achieve the target. By doing so, the disadvantage is the possible increase in aerothermal losses. Thus it is a tradeoff between temperature drop, pressure drop and aerothermal losses which has to compromise.
8. CONCLUSIONS

Numerical analyses are performed in this work to design internal cooling channels for effective heat transfer in the trailing edge region of gas turbine blades. Furthermore, to accommodate more channels in the chord-wise direction of the blade, the effect of a reduction in width of the inlet pass of the two-pass rectangular channel on heat transfer and pressure drop is numerically analysed. The last study is a design problem, where the conjugate heat transfer approach has been adopted. A design of a two-pass trapezoidal channel with internal cooling has been recommended after proposing and numerically evaluating a number of different configurations. This channel represents the trailing edge of a turbine blade, which is operated under engine-similar conditions. The ANSYS CFD package was used for the numerical computation of this work. The geometry and the computational mesh were produced in ANSYS ICEM-CFD and the solver was ANSYS FLUENT. The work can be divided mainly into three sections. The first section focuses on the rectangular cross-section two-pass channels, which represents the mid-chord region of the blade. The second section is about the trapezoidal cross-section two-pass channels, which represents the trailing edge of the blade. The third and final section uses conjugate heat transfer analysis to develop and evaluate new concepts for the design of a channel to enhance heat transfer at the trailing edge.

The heat transfer and pressure drop in smooth and ribbed rectangular two-pass channels with inlet pass aspect ratio $W_{in}/H = 1:3$ and outlet pass aspect ratio $W_{out}/H = 1:1$ were computed at different divider-to-tip wall distances ($W_{eff}/H$). The results were then compared to the results for a serpentine channel with an inlet aspect ratio of $W_{in}/H = 1:2$ and outlet pass aspect ratio of $W_{out}/H = 1:1$. For both smooth and ribbed channels, the relative pressure drop was found to decrease with an increase in divider-to-tip wall distance. It was also found that the decrease in aspect ratio of inlet pass results in decrease in pressure drop. There is no significant effect on the heat transfer in the inlet pass of smooth channels with a decrease in the aspect ratio while at bend and outlet pass, heat transfer decreases with the decrease in the aspect ratio of inlet pass. In case of ribbed channels, heat transfer at the tip wall as well as at the bend bottom decreases with a decrease in aspect ratio of inlet pass. Also it attains a constant value or increases for other surfaces with the increase in divider-to-tip wall distance. It is concluded from the results that a high aspect ratio inlet channel (1:2) is preferable to the low aspect ratio channel (1:3), if the purpose is to enhance heat transfer at outlet-pass and the bend regions but this channel offers a higher pressure drop.

Similar to rectangular channels, heat transfer and pressure drop in trapezoidal two-pass channel, which represents the trailing edge of the blade, was computed. The aim of the study was to compare three different designs for heat transfer enhancement at trailing edge without ejection openings. Different RANS models were tested and it was found that the low-Re $k$-$\varepsilon$ turbulence model predicts heat transfer much better than the other $k$-$\varepsilon$ or $k$-$\omega$ models in the outlet pass of a
two-pass trapezoidal channel. The comparison between the new designs for trailing edge heat transfer shows that a channel with ribs on the trailing edge is a better option only due to better thermal performance. If pressure losses are acceptable, the staggered arrangement is a recommended design to be used for the outlet pass heat transfer enhancement. The study shows that the rib height and pitch are important parameters in designing the ribbed channel. A wrong selection may cause resistance to the flow and a reduction in heat transfer.

Conjugate heat transfer analyses is helpful in calculating the metal temperatures, thus it can be applied where the design question is to limit the metal temperature. Such a study was performed where the aim of the design process was to bring the wall temperatures below a target value with a minimum drop in pressure in the trailing edge of a real engine vane operated under the real engine conditions. Steam was used as the coolant and its supremacy over air has been proven in this study. It was found that to obtain similar wall temperature levels; the requirement of steam flow rate is less compared to air e.g., to attain a wall temperature level obtained by using air, requirement of steam was found 45% less. This helps in reduction in the required pumping power. Nine basic designs of the two-pass trapezoidal channel were tested but none was able to achieve the wall temperature target. Different concepts were developed that are modifications were made to the basic case to achieve the target. The best arrangement to cool the trailing edge under given conditions is to provide a staggered arrangement of ribs in a channel with L-shaped tapered divider wall with a converging opening in it. But the thermal performance of such a channel is low due to the increase in pressure drop. The channel with tilted divider wall and staggered arrangement of ribs has better thermal performance but it has a comparatively larger area at the corner that is above the target value. Truncating the portion which is above the target value will increase the thickness of the trailing edge and can increase the aerothermal loss. Thus it is concluded that the best design of trailing edge which uses internal cooling is a tradeoff between heat transfer, pressure loss and aerothermal loss.
9. **FUTURE WORK**

Based on the results obtained from the research work performed in the present thesis, the following possibilities for future work are proposed.

**Rectangular Channel**

In the present thesis only two aspect ratios of the inlet pass are compared. In order to optimize the channel width at the inlet pass, the comparison range should be broadened. Similarly, the effect of change in width of the outlet pass was not included in the present thesis work. A wide range of variations in this can be helpful in optimizing the width of the outlet pass.

**Trapezoidal Channel**

The ribs on the bottom surface of the trapezoidal channel were orthogonal to the flow. It has been shown in the literature that ribs placed at an angle of 45° to the flow perform better; therefore such an arrangement of ribs in the trapezoidal channel can also be tested. Furthermore the ribs were arranged at the bottom wall of the channel only. Ribs at the tilted top wall can be added and its effect on heat transfer and pressure drop can be studied. The rib height and pitch at the trailing edge wall can be varied to an optimised value. Similar to the rectangular channel, divider-to-tip wall distance can be varied also to obtain an optimised value. The study was conducted at only one Reynolds number. Data can be obtained at different Reynolds numbers. Lastly, if the computational facility is available Large Eddy Simulations can be performed to minimize the difference between numerical and experimental results.

**Conjugate Heat Transfer**

The conjugate heat transfer analysis is a design problem. The CFD work has no experimental validation. Though this work helps in determining the best design amongst all, experimental validation of these results is still needed. The experiment can be performed for the best case to obtain the real data. In addition to that, CFD simulations can be used to further optimise the cooling channel like the angle of the tilted wall, placement of the opening in the divider wall, shape of the opening etc.
10. REFERENCES

Annerfeldt M. O.; Persson J. L.; Torisson T.; 2001

ANSYS FLUENT 12.0; Theory Guide

ANSYS FLUENT 12.0; User’s Guide

Arsenyev L.; Polyshchuk V.; Sokolov N.; 1994
“Application of Water/Steam for Cooling Gas Turbine Blades”, *Turbo design laboratory of St. Petersburg Technical University, Russia*.

Bell C.; Clarkson P. J.; Dawes W. N.; 2009
“The Design Of Turbine Blade Internal Cooling Systems”,
Available:http://wwwedc.eng.cam.ac.uk/research/processmanagement/pm3/turbinedesign/

Bohn, D.; Bonhoff B.; Schonenborn H.; Wilhelmi H.; 1995a
“Validation of a Numerical Model for the Coupled Simulation Fluid Flow and Diabatic Walls with Application to Film-cooled Gas Turbine Blades,” *VDI-Berichte No. 1186*.

Bohn D.; Bonhoff B.; Schonenborn H.; Wilhelmi H.; 1995b
“Prediction of the Film-cooling Effectiveness of Gas Turbine Blades Using a Numerical Model for the Coupled Simulation of Fluid Flow and Diabatic Walls,” *AIAA paper 95-7105*.

Bohn D.; Bonhoff B.; Schonenborn H.; 1995c

Bohn D.; Becker V.; Kusterer K; Otsuki Y.; Sugimoto T.; Tanaka R.; 1999

Borgnakke C.; Sonntag R. E.; 2009
Brundage A. L.; Plesniak M. W.; Lawless P. B.; Ramadhyani S.; 2007

Cakan M.; Taslim M. E.; 2007

Chandra P. R.; Han, J. C.; and Lau, S. C.; 1988

Chandra P. R.; Alexander C. R.; Han J. C.; 2003

Cho H. H.; Goldstein R. J.; 1995


Coletti F.; Armellini A.; Arts T.; Scholtes C.; 2011

Cravero C.; Giusto C.; Massardo A.F.; 1999

Cunha J. F.; 1994

Cunha J. F.; 2006
Ekkad S. V.; Pamula G.; Shatiniketanam M.; 2000

GE Energy; 2009
“H System*”
Last accessed: 7\textsuperscript{th} December 2010.

Gupta A.; SriHarsha V.; Prabhu S. V.; Vedula R. P.; 2008

Han J. C.; Glicksman L. R.; Rohsenhow W. M.; 1978

Han J. C.; 1984

Han J. C.; Park J. S.; 1988

Han J. C.; OU S.; Park J. S.; Lei C.K.; 1989

Han J. C.; Zhang Y. M.; and Lee C. P.; 1992

Han J.C.; Dutta S.; Ekkad S.V.; 2000

Huang Y.; Ekkad S. V.; Han; J. C.; 1998
Hwang J.J.; Lu C.C.; 2001

Iacovides H.; 2006
“Applications in Convective Heat Transfer with Emphasis on Blade Cooling Flows, Part-1, Background and Methodology”, Lecture Notes, Department of Mechanical Aerospace and Manufacturing Engineering, UMIST.

Incropera F.P.; De Witt D. P.; 2002,


Kiml R.; Mochizuki S.; Murata A.; 2001

Kiml R.; Mochizuki S.; Murata A.; Sulitka M.; 2003

Kirillov I. I.; Arsen’ev L.V.; 1986

Kulasekharan N.; Prasad B.; 2010

Kunstmann S.; Jens Von Wolfersdorf; Ruedel U.; 2009

Launier B. E.; Spalding D. B.; 1972
Lee S. W.; Ahn H. S.; Lau S. C.; 2007
“Heat (Mass) Transfer Distribution in a Two-Pass Trapezoidal Channel With a 180° Turn,”

Li X.; Gaddis J.L.; Wang T.; 2001

Louis J.F.; Hiraoka K.; Elmasri M.A.; 1983

MacDonald J.A.; 2003


Moon S. W.; Endley S.; Lau S. C.; 2002

Murata A.; Mochizuki S.; 2004

Obata M.; Yamaga J.; Taniguchi H.; 1989
"Heat Transfer Characteristics of A Return Flow Steam-Cooled Gas Turbine Blade”,

Okamura T.; Koga A.; Kawagishi H.; 2002
Pamula G.; Ekkad S. V.; Acharya S.; 2001


Polezhaev J; 1997

Power Engineering; 2009
“Steam cooling his the mark”

Rau, g.; Çakan, M.; Moeller, D.; Arts, T.; 1998

Rigby D. L.; Lepicovsky J.; 2001

Rubensdörffer F. G.; 2006

Sarvanamutto H. I. H.; Rogers G. F. C.; Cohen H.; 2005

Sewall E. A.; Tafti D. K.; 2005

Shih T.I.; Lin Y.L.; Stephens M.A.; 2001

Siddique W.; El-Gabry L.; Shevchuk I.V.; Fransson T.H.; 2011
“Validation and Analysis of Numerical Results for a Two-Pass Trapezoidal Channel with Different Cooling Configurations of Trailing Edge”, Accepted for publication in ASME Journal of Turbomachinery. Manuscript ID: Turbo-11-1168.

Singh O.; Yadav R.; 1995

Smith D.; 2004

Su G.; Chen H. C.; Han J.C.; Heidmann J. D.; 2004

Sundberg J.; 2006

Taslim M. E.; Li T.; Spring S. D.; 1995

Taslim M. E.; Li T.; Spring S. D.; 1997


Wang Y. Q.; Jackson, P. L.; Phaneuf T. J.; 2009

Wicks F.; Maleszewski J.; Wright C.; Zarybnicky J.; 2002
“Thermodynamic Analysis of an Enhanced Gas and Steam Cycle”, *IECEC Paper No. 20132*.

Wolfshtein M.; 1969

Wright L. M.; Fu W. L.; Han J. C.; 2004

Wright L.M.; Gohardani A. S.; 2008
APPENDIX A  GOVERNING EQUATIONS

The governing equations of fluid flow represent mathematical description of the conservation laws of physics. These includes

- Conservation of mass i.e., the mass of a fluid is conserved (Continuity equation)
- Conservation of momentum i.e., the rate of change of momentum equals the sum of the forces on a fluid particle (Newton’s second law)
- Conservation of energy i.e., the rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle (first law of thermodynamics)

A.1  Conservation of Mass
The general form of the mass conservation equation which is valid for incompressible as well as compressible flows is given by

$$ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 $$  \hspace{1cm} (A.1)

A.2  Conservation of Momentum
For fluid passing an infinitesimal control volume that is fixed in space, the general form of the momentum equation yield by conservation of momentum is given as

$$ \frac{\partial}{\partial t} \left( \rho \vec{V} \right) + \nabla \cdot \rho \vec{V} \vec{V} = \rho \vec{f} + \nabla \cdot \vec{\Pi} $$  \hspace{1cm} (A.2)

In the above equation, the term $\nabla \cdot \rho \vec{V} \vec{V}$ represents the rate at which momentum is lost by convection per unit volume from the control surface. $\vec{f}$ is the force per unit mass. $\vec{\Pi}$ represents the components of stress tensor which consists of normal and shear stresses and for Newtonian fluid (in which stress at a point is linearly dependent on the rate of strain of fluid) it is given by

$$ \vec{\Pi} = -p \delta_{ij} + \mu \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right] = -p \delta_{ij} + \tau_{ij} $$  \hspace{1cm} (A.3)

Where $\delta_{ij}$ is the Kronecker delta function ($\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$)
A.3 Conservation of Energy

The First Law of Thermodynamics is applied to the fluid passing an infinitesimal control volume fixed in space to obtain the energy equation.

\[
\frac{\partial E_t}{\partial t} + \nabla \cdot E_t \vec{V} = \frac{\partial Q}{\partial t} + \nabla \cdot k_{\text{eff}} \nabla T + \rho \vec{f} \cdot \vec{V} + \nabla \cdot \left( \Pi_{\beta \beta} \vec{V} \right) \tag{A.4}
\]

where \( E_t \) is the total energy per unit volume:

\[
E_t = \rho \left( e + \frac{V^2}{2} + \text{potential energy} + \cdots \right) \tag{A.5}
\]

And \( k_{\text{eff}} \) is the effective conductivity which includes turbulent thermal conductivity and is defined according to the turbulence model used. ANSYS FLUENT Theory guide [2009].
APPENDIX B  CFD AND TURBULENCE MODELLING

The governing equations of fluid flow representing the mathematical description of the conservation laws of physics were given in Appendix A and are called instantaneous or exact governing equations. These partial differential equations are coupled and can be used to predict the transportation of mass and energy of many fluid problems. For most engineering problems, the analytical solution of these equations is not possible. However, these equations can be discredited and solved numerically which is the subject of Computational Fluid Dynamics (CFD).

The approach of CFD is to replace the domain of interest which is continuous with a discrete domain using a grid. Thus, each flow variable which was previously defined at every point in the domain is now defined only at the grid points. For every flow variable, CFD solution would directly solve at the grid points and the values at other locations are determined by interpolating the values at the grid point.

ANSYS Fluent uses a control-volume-based method to discretize the governing equations of flow field. This method consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis [ANSYS FLUENT Theory guide; 2009].

B.1 Turbulence

Turbulent fluid motion is an irregular condition of flow in which the various quantities show a random variation with time and space. The random motion (eddies) in the flow increases with the increase in the Reynolds number of the flow. It also increases if there are tabulators in the flow path. These eddies are of varying length and time scales. Due to eddying motion in turbulent flows, the particles of fluid are brought closer together. This results in effective exchange of mass, heat and momentum. A turbulent flow is characterized by the mean value of the properties and the statistical properties of their fluctuations. For example for velocity components:

\[ u_i = \overline{u_i} + u'_i \quad (i = 1, 2, 3) \]  \hspace{1cm} (B.1)

where \( \overline{u} \) and \( u'_i \) are the mean and the fluctuating velocity components.

Similarly for scalar quantities \( \phi \) (like energy and pressure), the relationship is of the form

\[ \phi = \overline{\phi} + \phi' \] \hspace{1cm} (B.2)

B.2 Reynolds-Averaged Navier-Stokes (RANS)

For complex flows which are common in engineering problems, it is not possible to model the whole range of the turbulent eddies by Direct Numerical Simulation (DNS) with the available
computer resources. Large Eddy Simulation (LES) offers a solution to this problem such that it filters the governing equations to contain the large eddies and to model the smaller eddies only. This reduces the requirement of the computational resources on expense of the accuracy. Another common method used is to time-average the governing equations which results in the Reynolds Averaged Navier-Stokes (RANS) equations.

The Cartesian tensor form of the RANS equations is given by (the overbar on the mean velocity has been dropped)

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \]  
\[ \frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j}(-\rho u_i u_j') \]  

By time-averaging the instantaneous Navier-Stokes equation, additional terms appear that represents the effect of turbulence. These are the Reynolds stresses, \(-\rho u_i u_j'\), which should be modelled in order to close the Eq. B.3 and B.4 [ANSYS FLUENT Theory guide; 2009].

### B.3 Turbulence Modelling

The Reynolds averaging of the instantaneous Navier-Stokes equations results in unknown terms known as Reynolds stresses. In order to close the problem, the Reynolds stresses should be modelled and the process to do this is called turbulence modelling.

#### B.3.1 The Boussinesq Hypothesis

The Boussinesq hypothesis makes the assumption that the Reynolds stresses can be expressed in terms of mean velocity gradients. Thus the Reynolds stresses are given by:

\[-\rho u_i u_j' = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial u_k}{\partial x_k} \right) \delta_{ij} \]  

where \(\mu_t\) is the turbulent or eddy viscosity. \(k\) is the turbulent kinetic energy and is defined in terms of the velocity fluctuations in each of the three coordinate directions:

\[ k = \frac{1}{2} \left( (u')^2 + (v')^2 + (w')^2 \right) \]
Depending on the turbulence model, the eddy viscosity is calculated in a different manner. These models can be grouped in 0-, 1-, 2-equation models. Once the eddy viscosity is calculated, it can be used to calculate the value for the Reynolds stress, which than can be used for the calculation of the momentum equation. All turbulence models use approximations to accomplish this goal, and it is the nature of the flow conditions in each specific application, that determines which set of approximations is acceptable for use [ANSYS FLUENT Theory guide; 2009].

B.3.2 Two Equation Turbulence Models

The two equation turbulence models solve two transport equations to represent the turbulent properties and get the eddy viscosity. This allows the model to account for history effects like convection and diffusion of the turbulent energy. One of the transported variables is the turbulent kinetic energy, $k$, while the second transported variable depends upon the type of two equation model. It can be the turbulent dissipation $\varepsilon$ in case of the k-\(\varepsilon\) model or the specific dissipation $\omega$ in case of k-\(\omega\) model. The first transport variable determines the energy in turbulence while the second variable determines the scale of the turbulence. Different types of two equation turbulence models used in this thesis work include:

B.3.2.1 the Low-Re k-\(\varepsilon\) Model

There are a number of different low-Re k-\(\varepsilon\) models provided by ANSYS FLUENT but the one used in this thesis work was proposed by Launder and Spalding [1972]. It is a robust model, meaning that it is computationally stable, even in the presence of other, more complex physics. It is applicable to a wide variety of turbulent flows. It is semi-empirical, based to a large part on observations of mostly high Reynolds Number flows, and thus restricting its applicability to flows far from the influence of boundaries. But later extensions have been introduced to allow for wall proximity effects.

The two transport equations that need to be solved for this model are for the kinetic energy of turbulence, $k$, and the rate of dissipation of turbulence, $\varepsilon$. The model transport equation for $k$ is derived from the exact equation; while the model transport equation for $\varepsilon$ was obtain using physical reasoning.

To summarize the solution process for the k-\(\varepsilon\) model, transport equations are solved for the turbulent kinetic energy and the dissipation rate. The solutions for $k$ and $\varepsilon$ are used to compute the turbulent viscosity $\mu_t$. Using the results for $\mu_t$ and $k$, the Reynolds stresses can be computed for substitution into the momentum conservation equation. Once the momentum equations have been solved, the new velocity components are used to update the turbulence generation term and the process is repeated [ANSYS FLUENT Theory guide; 2009].
In these equations, $G_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients. $G_b$ is the generation of turbulence kinetic energy due to buoyancy. $Y_M$ represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$ respectively. $S_k$ and $S_\varepsilon$ are user-defined source terms. The constants are $C_{1\varepsilon}=1.44$, $C_{2\varepsilon}=1.92$, $C_\mu=0.09$, $\sigma_k=1.0$ and $\sigma_\varepsilon=1.3$.

For all $k$-$\varepsilon$ models, the turbulent viscosity is derived from both $k$ and $\varepsilon$. The difference involves a term $C_\mu$, which has a value of 0.09 in case of low-Re $k$-$\varepsilon$ model.

$$\mu_i = \rho C_\mu \frac{k^2}{\varepsilon} \quad (B.9)$$

**B.3.2.2 the RNG k- $\varepsilon$ Model**

A rigorous statistical technique called renormalization group theory is used to derive the RNG $k$-$\varepsilon$ model. In comparison with low-Re $k$-$\varepsilon$ model (Launder and Spalding), RNG $k$-$\varepsilon$ model has an additional that improves the accuracy for rapidly strained flows. Also this model provides as analytical formula for turbulent Prandtl numbers. As the model is derived using a mathematical technique called “normalization group” methods, therefore the constants differs from those in the low-Re model.

$k$ -equation:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_l}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (B.10)$$
ε-equation:
\[
\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \alpha_\varepsilon \mu_{\varepsilon} \frac{\partial\varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon - R_\varepsilon \tag{B.11}
\]

In these equations, the additional terms \(\alpha_k\) and \(\alpha_\varepsilon\) are the inverse effective Prandtl numbers for \(k\) and \(\varepsilon\) respectively. The turbulent viscosity (eddy viscosity) is computed by Eq. B.9. The constants are \(C_{1\varepsilon} = 1.42,\ C_{2\varepsilon} = 1.68\) and \(C_\mu = 0.0845\) [ANSYS FLUENT Theory guide; 2009].

B.3.2.3 the Realizable k-ε Model

The term “realizable” means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. The realizable k-ε model has a new formulation for the turbulent viscosity and a new transport equation for dissipation rate. The transport equation for dissipation rate is derived from the exact equation for the transport of the mean-square vorticity fluctuation [ANSYS FLUENT Theory guide; 2009].

\(k\)-equation:
\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_k}{\sigma_k} \right] \frac{\partial k}{\partial x_j} + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{B.12}
\]

ε-equation:
\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_\varepsilon}{\sigma_\varepsilon} \right] \frac{\partial \varepsilon}{\partial x_j} + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{2\varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{\sqrt{k}} C_{3\varepsilon} G_b + S_\varepsilon \tag{B.13}
\]

where
\[
C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = \frac{k}{S_\varepsilon}, \quad S = \sqrt{2S_y S_y}
\tag{B.14}
\]

The eddy viscosity is computed from Eq. B.9 but \(C_\mu\) is no more a constant. It is computed from
\[
C_\mu = \frac{1}{A_v + A_s \frac{k U^*}{\varepsilon}} \tag{B.15}
\]

where
\[
U^* = \sqrt{S_y S_y + \tilde{O}_y \tilde{O}_y}
\tag{B.16}
\[
\widetilde{\Omega}_{ij} = \Omega_{ij} - 2 \varepsilon_{ijk} \omega_k \quad \text{(B.17)}
\]
\[
\Omega_{ij} = \Omega_{ij} - \varepsilon_{ijk} \omega_k \quad \text{(B.18)}
\]

Here \( \widetilde{\Omega}_{ij} \) is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity \( \omega_k \). The model constants are

\[
A_o = 4.04, \ A_x = \sqrt[3]{6 \cos \phi} \quad \text{(B.19)}
\]

\[
\phi = \frac{1}{3} \cos^{-1}\left(\sqrt{6}W\right), \ W = \frac{S_{ij} S_{jk} S_{ki}}{S^3}, \widetilde{S} = \sqrt{S_{ij} S_{ij}}, \ S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \quad \text{(B.20)}
\]

\( C_\mu \) can be shown to recover the standard value of 0.09 for an inertial sublayer in an equilibrium boundary layer [ANSYS FLUENT Theory guide; 2009].

**B.3.2.4 the Low-Re \( k-\omega \) model**

In this model the low-Re \( k \) equation is solved, but as a length determining equation, \( \omega \) is used. This quantity is often called specific dissipation from its definition

\[
\omega = -\frac{\varepsilon}{k} \quad \text{(B.21)}
\]

The two transport equations that need to be solved for this model are

**\( k \)-equation:**

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad \text{(B.22)}
\]

**\( \omega \)-equation:**

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega \quad \text{(B.23)}
\]

In these equations, \( G_k \) represents the generation of turbulence kinetics energy due to mean velocity gradients. \( G_\omega \) represents the generation of \( S_k \) and \( S_\omega \) are user-defines source terms.

The turbulent viscosity is
The coefficient $\alpha^*$ dumps the turbulence viscosity causing a low-Reynolds number correction. It is given by

$$\alpha^* = \alpha^*_\infty \left( \frac{\alpha_0^* \left( \frac{\text{Re}_i}{\text{Re}_k} \right)}{1 + \frac{\text{Re}_i}{\text{Re}_k}} \right)$$

(B.25)

where

$$\text{Re}_i = \frac{\rho k}{\mu \omega}$$

(B.26)

$$\text{Re}_k = 6$$

(B.27)

$$\alpha_0^* = \frac{\beta_i}{3}$$

(B.28)

$$\beta_i = 0.072$$

(B.29)

In high-Reynolds-number form of the $k$-$\omega$ model,

$$\alpha^* = \alpha_0^* = 1$$

(B.30)

The rate of dissipation of turbulence is

$$\varepsilon = \beta^* \omega k$$

(B.31)

### B.3.2.5 Shear-Stress Transport (SST) $k$-$\omega$ model

The Shear-Stress Transport (SST) $k$-$\omega$ model includes some refinements in Wilcox's $k$-$\omega$ model. The $k$-$\omega$ model is converted into a $k$-$\omega$ formulation and is called transformed $k$-$\omega$ model. The Wilcox's $k$-$\omega$ model and the transformed $k$-$\omega$ model are both multiplied by a blending function and then added together. The blending function is one in the near wall region, which activates the Wilcox's $k$-$\omega$ model and it is zero away from wall, which activates the transformed $k$-$\omega$ model. SST $k$-$\omega$ model is more accurate and reliable for adverse pressure gradient flows, airfoils and transonic shock waves [ANSYS FLUENT Theory guide; 2009].

The two transport equations that need to be solved for this model are:
The quantity $G_k$ is the production of turbulence kinetic energy and is defined as:

$$G_k = \min(G_k, 10 \beta k \omega) \quad (B.34)$$

The quantity $G_\omega$ represents the production of $\omega$ and is given by:

$$G_\omega = \frac{\alpha}{\nu_t} G_k \quad (B.35)$$

$S_k$ and $S_\omega$ are user-defined source terms. $D_{\omega}$ represents the cross-diffusion term and is defined as

$$D_{\omega} = 2(1 - F_1) \rho \sigma_{\omega,2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (B.36)$$

Where

$$F_1 = \tanh \left( \Phi_1^4 \right) \quad (B.37)$$

$$\Phi_1 = \min \left[ \max \left( \frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right), \frac{4 \rho k}{1.168 D_{\omega}^* y^2} \right] \quad (B.38)$$

$$D_{\omega}^* = \max \left[ 2 \rho \frac{1}{1.168 \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right] \quad (B.39)$$

where $y$ is the distance to the next surface.

### B.4 Wall Treatment

In most cases the turbulent flows are influenced by adjacent walls. An example of this are flows in a channel, around an airfoil and the flow of rivers. The walls have two major effects:

- They damp the wall normal components which makes the turbulent flow anisotropic.
- They increase the production of turbulence through the shearing mechanism in the flow. When a fluid flows along the wall, a boundary layer is created. Inside this layer, the velocity gradient varies from zero at the wall to its free stream value away from wall. In case of heat
transfer applications, there also exists a thermal boundary layer. The thermal as well as the velocity gradients are strongest in the near wall region and both heat transfer and friction are computed using these gradients. Therefore it is important to capture these near wall gradients as exact as possible.

The near wall region can be divided into three layers. The innermost layer is called viscous sub-layer. The flow in this layer is almost laminar and the molecular viscosity plays a dominant role in momentum and heat transfer. The outer layer is called fully turbulent layer where turbulence plays a significant role. In between the viscous sub-layer and fully turbulent layer, an interim region called buffer layer exists. The effects of molecular viscosity and turbulence are equally important in this region [ANSYS FLUENT Theory guide; 2009]. Figure B.1 illustrates these subdivisions of the near-wall region, plotted in semi-log coordinates.

In Fig. B.1, $y^+$ is the non-dimensional distance from the wall and is given by

$$y^+ = \frac{\rho U_\tau y}{\mu} \quad (B.40)$$
B.4.1 Wall Calculation Methods

For on-design simulations without any large separated regions it is often sufficient to use a wall-function model close to the wall, preferably with some form of non-equilibrium wall-function that is sensitized to streamwise pressure gradients.

For off-design simulation, or simulations involving complex secondary flows and separations, it is often necessary to use a low-Re model. There exist many low-Re models that have been used with success in turbomachinery simulations. A robust and often good choice is to use a one-equation model. There are also several Low-Re k-ε models that work well.

There are two methods to capture the near wall variations. One is a standard method, where a very fine mesh is applied near the wall. This method is called the integration method or near-wall modelling approach. It is necessary then to use a LRN (Low Reynolds Number) type of turbulence model. Another method is to use semi-empirical formulas called wall functions. In this approach, the viscosity affected region (viscous sub-layer and buffer layer) is not resolved. The wall functions bridges the viscosity-affected region between the wall and the fully-turbulent region. HRN (High Reynolds Number) turbulence models are used in this approach. In high Reynolds Number flows, the viscosity affected near wall region does not need to be resolved. Therefore, the wall function approach saves computational recourses, substantially. But in low Reynolds Number flows, the near wall model approach is more appropriate to use [ANSYS FLUENT Theory guide; 2009]. Figure B-2 schematically shows the two methods.

Wall functions comprises of

- Law of wall for mean velocity and temperature.
- Formulas for near wall turbulent quantities.

**Figure B-2:** Near Wall treatments in ANSYS FLUENT [ANSYS FLUENT Theory guide; 2009]
B.4.2 Limitations of Wall Function Approach

When a flow is bounded by a wall and is of high Reynolds Number, the wall function approach predicts reasonably accurate. But there are cases when it becomes less reliable. Examples of such cases are:

- Highly viscous, low velocity flows
- Blowing or suction through the wall, like in film cooling of an airfoil
- High pressure gradients which lead to boundary layer separation
- Flow near rotating disks and buoyancy driven flows where strong body forces exists

For such simulation cases, the near-wall modelling approach has to be combined with mesh resolution in near wall region. ANSYS FLUENT provides the enhanced wall treatment for such cases [ANSYS FLUENT Theory guide; 2009].

B.4.3 Enhanced Wall Treatment

In ANSYS FLUENT, the enhanced wall treatment is a near-wall modelling method that combines a two-layer model with so-called enhanced wall function. The two-layer approach is 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. In the fully turbulent region, the k-$\varepsilon$ models are employed while in the viscosity-affected near-wall region, the one-equation model of Wolfstein [1969] is employed. The mesh should be fine enough to be able to resolve the viscous sublayer (typically with the first near-wall node placed at $y^+ = 1$). This limitation imposes a large computational requirement [ANSYS FLUENT Theory guide; 2009].