The Pennsylvania State University

The Graduate School

Department of Aerospace Engineering

ENDWALL SHAPE MODIFICATION USING VORTEX GENERATORS AND FENCES
TO IMPROVE GAS TURBINE COOLING AND EFFECTIVENESS

A Dissertation in
Aerospace Engineering
by
Zeki Özgür Gökçe

© 2012 Zeki Özgür Gökçe

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Doctor of Philosophy

December 2012
The dissertation of Zeki Özgür Gökçe was reviewed and approved* by the following:

Cengiz Camcı  
Professor of Aerospace Engineering  
Dissertation Advisor  
Chair of Committee

Sven Schmitz  
Assistant Professor of Aerospace Engineering

Savaş Yavuzkurt  
Professor of Mechanical Engineering

Timothy Miller  
Senior Research Associate, Applied Research Laboratory

George A. Lesieutre  
Professor of Aerospace Engineering  
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School
ABSTRACT

The gas turbine is one of the most important parts of the air-breathing jet engine. Hence, improving its efficiency and rendering it operable under high temperatures are constant goals for the aerospace industry. Two types of flow within the gas turbine are of critical relevance: The flow around the first row of stator blades (also known as the nozzle guide vane blade - NGV) and the cooling flow inside the turbine blade cooling channel. The behavior of the former flow type affects the total pressure level downstream of the NGV, thereby affecting the efficiency of the gas turbine. The behavior of the latter flow type affects the cooling performance of turbine blades, having a direct effect on the maximum total temperature the turbine material can withstand, as well as affecting the level of thrust produced via the total temperature. The flow in the vicinity of the turbine blades and the endwall boundary layer has a great effect on the behavior of the overall flow through the gas turbine. The flow near the pin-fins contained inside turbine blade cooling channels dictates the cooling performance of the blade. These two facts have prompted the aerospace industry to investigate the potential benefits of modifying the shape of the endwall. The results of various studies showed improvements in both flow types. Following this thought process, this thesis study focused on finding innovative ways of attaining even higher performance. We previously demonstrated that adding upstream endwall fences leads to beneficial changes in the flow near the NGV blade. Inspired by our prior findings, we decided to analyze the effects of endwall shape modifications on turbine cooling channel flow, in addition to the flow near the NGV. In short, the subject of this thesis work was to search for methods that could improve the characteristics of these two types of flows, thus enabling superior engine performance. The innovative aspect of our work was to apply an endwall shape modification previously employed by non-aerospace industries for cooling applications, to the gas turbine cooling flow which is vital to aerospace propulsion.
Since the costs of investigating the possible benefits of any idea via extensive experiments could be quite high, we decided to use computational fluid dynamics (CFD) followed by experimentation as our methodology. We decided to analyze the potential benefits of using vortex generators (VGs) as well as the rectangular endwall fence. Since the pin-fins used in cooling flow are circular cylinders, and since the boundary layer flow is mainly characterized by the leading edge diameter of the NGV blade, we modeled both the pin-fins and the NGV blade as vertical circular cylinders. The baseline case consisted of the cylinder(s) being subjected to cross flow and a certain amount of freestream turbulence. The modifications we made on the endwall consisted of rectangular fences. In the case of the cooling flow, we used triangular shaped, common flow up oriented, delta winglet type vortex generators as well as rectangular endwall fences. The channel contained singular cylinders as well as staggered rows of multiple cylinders. For the NGV flow, a rectangular endwall fence and a singular cylinder were utilized.

Using extensive CFD modeling and analysis, we confirmed that placing a rectangular endwall fence upstream of the cylinder created additional turbulent mixing in the domain. This led to increased mixing of the cooler flow in the freestream and the hotter flow near the endwall. As a result, we showed that adding a rectangular fence created a 10% mean heat transfer increase downstream of the cylinder.

When vortex generators are used, as the flow passes over the sharp edges of the vortex generators, it separates and continues downstream in a rolling, helical pattern. Combined with the effect generated by the orientation of the vortex generators, this flow structure mixes the higher momentum fluid in the freestream with lower momentum fluid in the boundary layer. Similar turbulent mixing behavior is observed over the entire domain, near the cylinders and the side walls. As a result, the heat transfer levels over the wall surfaces are increased and improved cooling is achieved. The improvements in heat transfer are obtained at the expense of acceptable pressure losses across the cooling channel. When the vortex generators are used, the CFD
modeling studies showed that overall heat transfer improvements as high as 27% compared to the baseline case are observed inside a domain containing multiple rows of cylinders. A price in the form of 13% pressure loss increase across the channel is paid for the heat transfer benefits. Experiments conducted in the open loop wind tunnel of the Turbomachinery Aero-Heat Transfer Laboratory of the Department of Aerospace Engineering of Penn State University supported the general positive trend of these findings, with a 14% overall increase in heat transfer over the constant heat flux surface when vortex generators are installed, accompanied by an 8% increase in pressure loss.

To our knowledge, our study is the first to demonstrate the positive effects of vortex generators used in conjunction with circular pin-fins on the heat transfer properties of gas turbine blade cooling channel flow. The findings of our study may also have practical implications for other scientific and industrial fields using flows of similar Reynolds numbers.
# TABLE OF CONTENTS

LIST OF FIGURES ix

LIST OF TABLES xviii

LIST OF VIDEOS xix

NOMENCLATURE xx

ACKNOWLEDGEMENTS xxiii

Chapter 1 Introduction 24

Chapter 2 Literature Review: Past Studies focusing on Heat Transfer and Performance Increase in Channel Flow and Turbine Flow 33

Chapter 3 Numerical Analysis to Obtain Increased Heat Transfer and Performance in Channel Flow and Turbine Flow 38

3.1 Theoretical Background: The \( k-\omega \) SST Turbulence Model 38

3.2 Details of the Computations and Flow Conditions 41

3.2.1 Fence Installed Cases 42

3.2.2 Vortex Generator Installed Cases. 44

3.3 Grid Independence Study 45

3.4 Numerical Steps of Attaining Performance Increase using Fence Related Effects 47

3.4.1 Geometrical Setup: Baseline Case 47

3.4.2 Mesh Structure: Baseline Case 50

3.4.3 Geometrical Setup: Fence Installed Case 52

3.4.4 Mesh Structure: Fence Installed Case 55

3.5 Numerical Steps of Attaining Heat Transfer Increase using Vortex Generator Induced Effects 57

3.5.1 Geometrical Setup: Baseline Case 57

3.5.2 Mesh Structure: Baseline Case 59

3.5.3 Geometrical Setup: Half Height Vortex Generator Installed Case 61

3.5.4 Mesh Structure: Half Height Vortex Generator Installed Case 64
Chapter 4 Numerical Validation of the Vortex Generator Concept

4.1 General Information on Vortex Generators

4.2 Numerical Validation
   4.2.1 Numerical Method
   4.2.2 Geometrical Setup
   4.2.3 Mesh Structure

4.3 The Half Height Vortex Generator Concept

4.4 Re = 2000 Results
   4.4.1 Two and Three Dimensional Streamlines
   4.4.2 Static Pressure Contours over the Bottom Surface
   4.4.3 Heat Transfer Coefficient Contours over the Bottom and Top Surface
   4.4.4 Heat Transfer and Pressure Change Results

4.5 Re = 500 Results
   4.5.1 Two and Three Dimensional Streamlines
   4.5.2 Heat Transfer Coefficient Contours over the Bottom Surface
   4.5.3 Heat Transfer and Pressure Change Results

4.6 Conclusions of the Validation Study

Chapter 5 Experimental Procedure for the Validation of Numerically Obtained Heat Transfer Improvements

5.1 Experimental Setup

5.2 Details of the Test Section

5.3 Details of the Experimental Procedure

Chapter 6 Results of the Computational Study for determining Performance Increase due to Fence Related Effects

6.1 Two and Three Dimensional Streamlines

6.2 Total Pressure Contours

6.3 Turbulent Kinetic Energy Contours

6.4 Heat Transfer Coefficient Contours over the Hub

6.5 Spanwise Total Pressure Distribution

6.6 Heat Transfer Coefficient Levels

6.7 Mass Averaged Total Pressure based C_p Analysis
Chapter 7 Results of the Computational and Experimental Study for determining Heat Transfer Increase due to Vortex Generator Related Effects

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.1 Reynolds Number Based Computational Fluid Dynamics Analysis</td>
<td>147</td>
</tr>
<tr>
<td>7.1.1 Two Dimensional Streamlines</td>
<td>148</td>
</tr>
<tr>
<td>7.1.2 Three Dimensional Streamlines</td>
<td>153</td>
</tr>
<tr>
<td>7.1.3 Heat Transfer Coefficient Contours on the Bottom and Top Surfaces</td>
<td>157</td>
</tr>
<tr>
<td>7.1.4 Heat Transfer and Pressure Change Results</td>
<td>167</td>
</tr>
<tr>
<td>7.2 Detailed Computational Fluid Dynamics Results for Re = 30000</td>
<td>174</td>
</tr>
<tr>
<td>7.2.1 Two Dimensional Streamlines</td>
<td>174</td>
</tr>
<tr>
<td>7.2.2 Three Dimensional Streamlines</td>
<td>175</td>
</tr>
<tr>
<td>7.2.3 Static Pressure Contours over the Bottom Surface</td>
<td>179</td>
</tr>
<tr>
<td>7.2.4 Heat Transfer Coefficient Contours over the Bottom and Top Surfaces</td>
<td>182</td>
</tr>
<tr>
<td>7.3 Comparison of the Baseline and Vortex Generator Installed Cases with the Plain Channel Case</td>
<td>185</td>
</tr>
<tr>
<td>7.4 Experimental Results</td>
<td>186</td>
</tr>
<tr>
<td>7.4.1 Conduction Loss Analysis</td>
<td>186</td>
</tr>
<tr>
<td>7.4.2 Adiabatic Wall Temperature Calculation</td>
<td>188</td>
</tr>
<tr>
<td>7.4.3 Temperature Distribution Analysis</td>
<td>189</td>
</tr>
<tr>
<td>7.4.4 Overall Heat Transfer Coefficient Distribution Analysis</td>
<td>192</td>
</tr>
<tr>
<td>7.4.5 Local Heat Transfer Coefficient Analysis</td>
<td>194</td>
</tr>
<tr>
<td>7.5 Explanation of the Underlying Physics related to the Vortex Generator Concept</td>
<td>201</td>
</tr>
</tbody>
</table>

Chapter 8 Conclusion

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.1 Endwall Fence Related Improvements</td>
<td>207</td>
</tr>
<tr>
<td>8.2 Vortex Generator Related Improvements</td>
<td>208</td>
</tr>
<tr>
<td>8.3 Recommendations for Future Studies</td>
<td>210</td>
</tr>
</tbody>
</table>

References

Appendix A: Experimental Uncertainty Analysis
LIST OF FIGURES

Figure 1.1 Internal layout of the gas turbine cooling channel, pin-fin cooling region circled in red 25

Figure 3.1 The static pressure coefficient calculated along the mid-height circumference of the middle cylinder of the first row 46

Figure 3.2 Three dimensional overview of the baseline case 48

Figure 3.3 Certain dimensions of the baseline case, all of which also apply to the fence installed case 49

Figure 3.4 Side view of the baseline case demonstrated in the x-z plane 49

Figure 3.5 The dense mesh structure near the cylinder and in the boundary layer 50

Figure 3.6 The gradually decreasing mesh density near the top of and at the inlet of the domain 51

Figure 3.7 The gradually decreasing mesh density as we move towards the outlet 51

Figure 3.8 Three dimensional overview of the fence installed case 52

Figure 3.9 Certain dimensions of the fence-installed case 53

Figure 3.10 Cross section details of the fence 53

Figure 3.11 Side view of the fence-installed case demonstrated in the x-z plane 54

Figure 3.12 Side views of each case demonstrated together for comparison 54

Figure 3.13 The dense mesh structure near the fence, cylinder and in the boundary layer 55

Figure 3.14 The gradually decreasing mesh density near the top of and at the inlet of the domain 56

Figure 3.15 The gradually decreasing mesh density as we move towards the outlet 56

Figure 3.16 Overview of the baseline case’s domain 58

Figure 3.17 Top view detailing the positions and dimensions of the staggered cylinder array 58
Figure 3.18 Overview of the baseline case’s mesh

Figure 3.19 A volume near the cylinder depicting the mesh structure

Figure 3.20 Another volume near the cylinder depicting the mesh structure.

Figure 3.21 The grid structure downstream of the staggered rows of cylinders

Figure 3.22 The gradually decreasing mesh density as we move towards the outlet

Figure 3.23 Top view of the half height vortex generator installed case’s domain

Figure 3.24 Side view of the half height vortex generator installed case’s domain

Figure 3.25 Overview of the baseline case’s mesh

Figure 3.26 Closer look at the volumes in the vicinity of the vortex generators and cylinders

Figure 3.27 Another look at the volumes in the vicinity of the vortex generators and cylinders

Figure 3.28 Volumes directly above the vortex generators

Figure 4.1 The delta wing type vortex generator

Figure 4.2 Common VG geometries

Figure 4.3 Common flow up

Figure 4.4 Common flow down

Figure 4.5 Dimensional information and layout of Torii et al.’s setup

Figure 4.6 Torii et al.’s heat transfer and total pressure loss data for the staggered array of cylinders

Figure 4.7 Geometrical layout of the baseline case (figure not to scale in the z-direction for enabling easier visualization)

Figure 4.8 Geometrical layout of the baseline case seen from the top

Figure 4.9 Geometrical layout of the vortex generator installed case (figure not to scale in the z-direction for enabling easier visualization).

Figure 4.10 Certain details of the vortex generators seen from the top
Figure 4.11 Overview of the baseline and vortex generator installed cases seen together for comparison (figure not to scale in the z-direction for enabling easier visualization) 79

Figure 4.12 Overview of the baseline case’s mesh (figure not to scale in the z-direction for enabling easier visualization) 80

Figure 4.13 Overview of the vortex generator installed case’s mesh (figure not to scale in the z-direction for enabling easier visualization) 80

Figure 4.14 The volumes which form the baseline case’s geometry 81

Figure 4.15 A face mesh which demonstrates the densely placed boundary layer nodes near the upper and lower surfaces 81

Figure 4.16 The placement of the nodes near to and away from the cylinder surfaces 82

Figure 4.17 Closer look at the hexahedral cells inside one particular volume 82

Figure 4.18 The volumes which form the vortex generator installed case’s geometry 83

Figure 4.19 Top view of the vortex generator installed case 84

Figure 4.20 Closer look at the inlet of the domain. 84

Figure 4.21 Closer look at the region encircled in red in Figure 4.20 85

Figure 4.22 Closer look at the volumes in the vicinity of the vortex generators 85

Figure 4.23 The volumes downstream of the vortex generators which are very similar to their baseline counterparts 86

Figure 4.24 Certain volumes which are highlighted to demonstrate the mesh structure 86

Figure 4.25 Closer look at a specific volume which lies between a pair of vortex generators 87

Figure 4.26 A volume adjacent to a cylinder 87

Figure 4.27 The densely spaced nodes near the top and bottom surfaces will provide good flow resolution 88
Figure 4.28 Geometrical layout of the half height vortex generator installed case (all dimensions are the same as the full height vortex generator installed case, except for the height of the half height VG)

Figure 4.29 Overview of the half height vortex generator installed case’s mesh

Figure 4.30 Close look at the mesh structure near one of the half height vortex generators

Figure 4.31 One of the half height vortex generators and some of the volumes near the VG highlighted

Figure 4.32 Streamlines in the streamwise plane of symmetry for the baseline case

Figure 4.33 Streamlines in the streamwise plane of symmetry for the full height vortex generator installed case

Figure 4.34 Comparison of the two dimensional streamlines for all cases

Figure 4.35 Three dimensional streamlines of the baseline case

Figure 4.36 Three dimensional streamlines of the full height vortex generator installed case

Figure 4.37 Comparison of the three dimensional streamlines for all cases

Figure 4.38 Static Pressure contours over the bottom surface for the baseline case

Figure 4.39 Static Pressure contours over the bottom surface for the full height vortex generator installed case

Figure 4.40 Comparison of the static pressure contours along the bottom surface: Baseline, full VG and half VG, respectively

Figure 4.41 Heat transfer coefficient contours for the baseline case

Figure 4.42 Heat transfer coefficient contours for the full height vortex generator installed case

Figure 4.43 Heat transfer coefficient contours for the baseline case - top surface

Figure 4.44 Heat transfer coefficient contours for the full height vortex generator installed case - top surface

Figure 4.45 Heat transfer coefficient contours for the half height vortex generator installed case - bottom surface

Figure 4.46 Heat transfer coefficient contours for the half height vortex generator installed case - top surface
Figure 4.47 Comparison of heat transfer contours over the bottom surface of each case: Baseline, full VG and half VG, respectively

Figure 4.48 Static pressure change calculation line and part of the heat transfer calculation regions

Figure 4.49 Comparison of the two dimensional streamlines

Figure 4.50 Comparison of the three dimensional streamlines

Figure 4.51 Comparison of heat transfer coefficient contours over the bottom surface

Figure 5.1 External view of the experimental facility

Figure 5.2 Schematic of the test section

Figure 5.3 External view of the plenum chamber, circular nozzle and converging duct

Figure 5.4 Schematic of the converging duct

Figure 5.5 Schematic of the diverging duct

Figure 5.6 The cylinders, semi-cylinders and vortex generators painted in black

Figure 5.7 The thermofoil heater

Figure 5.8 The test section covered with depron, including the portion below the thermofoil heater

Figure 5.9 The test section surface painted in black

Figure 5.10 Thermocouple between the depron and plexiglass

Figure 5.11 Intermediate thermocouple installation step

Figure 5.12 Thermocouple between the heater and depron

Figure 5.13 Final look of the conduction loss thermocouple setup

Figure 5.14 Camera calibration thermocouple

Figure 5.15 Conduction loss thermocouple plugs

Figure 5.16 Calibration thermocouple plug and the heater power cable

Figure 5.17 The plexiglass window

Figure 5.18 The infrared window secured on the plexiglass
Figure 5.19 The thermocouple probe and the Pitot velocity probe 126
Figure 5.20 The thermometer datalogger 127
Figure 5.21 The pressure measurement device used for measuring velocity and static pressure 127
Figure 5.22 Infrared camera 128
Figure 5.23 The final look of the test section when the cylinders are installed 129
Figure 5.24 The final look of the test section when the cylinders and vortex generators are installed 129
Figure 5.25 Final look of the experimental setup 130
Figure 6.1 Baseline case: Streamlines within the streamwise plane of symmetry 134
Figure 6.2 Fence-installed case: Streamlines within the streamwise plane of symmetry 134
Figure 6.3 The difference between total pressure values for the two cases, in the spanwise direction 141
Figure 6.4 The convective heat transfer coefficient magnitude over the hub along the streamwise axis, plotted for the two cases 143
Figure 6.5 The percentage change in h when a fence is employed, when compared to the baseline case 143
Figure 6.6 Mass averaged total pressure based Cp values over entire y-z planes 145
Figure 6.7 Mass averaged total pressure based Cp values over 2 mm-high rectangular planes extending above the hub surface 145
Figure 7.1 Re = 17000, Baseline and Vortex Generator installed cases, respectively 148
Figure 7.2 Re = 30000, Baseline and Vortex Generator installed cases, respectively 149
Figure 7.3 Re = 61000, Baseline and Vortex Generator installed cases, respectively 150
Figure 7.4 Re = 83000, Baseline and Vortex Generator installed cases, respectively 151
Figure 7.5 Re = 17000, Baseline and Vortex Generator installed cases, respectively 153
Figure 7.6 Re = 30000, Baseline and Vortex Generator installed cases, respectively 154
Figure 7.7 Re = 61000, Baseline and Vortex Generator installed cases, respectively 155
Figure 7.8 Re = 83000, Baseline and Vortex Generator installed cases, respectively

Figure 7.9 Re = 17000, Bottom Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.10 Re = 30000, Bottom Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.11 Re = 61000, Bottom Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.12 Re = 83000, Bottom Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.13 Re = 17000, Top Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.14 Re = 30000, Top Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.15 Re = 61000, Top Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.16 Re = 83000, Top Surface, Baseline and Vortex Generator installed cases, respectively

Figure 7.17 Top view of the bottom local and large areas

Figure 7.18 Isometric view of all the local areas

Figure 7.19 Isometric view of all the large areas

Figure 7.20 Overall (Sum of all large surfaces and cylinders) heat transfer increase compared to the baseline case when VGs are installed

Figure 7.21 Area Averaged Total Pressure Loss Coefficient variation

Figure 7.22 Baseline Case - Two dimensional streamlines in the streamwise plane of symmetry

Figure 7.23 Vortex Generator Installed Case - Two dimensional streamlines in the streamwise plane of symmetry

Figure 7.24 Baseline Case - Streamlines originating from rakes close to the lower surface

Figure 7.25 Vortex Generator Installed Case - Streamlines originating from rakes close to the lower surface
Figure 7.26 Baseline Case - Streamlines originating from rakes close to the upper surface

Figure 7.27 Vortex Generator Installed Case - Streamlines originating from rakes close to the upper surface

Figure 7.28 Baseline Case - Streamlines originating from within the wake of the first row

Figure 7.29 Vortex Generator Installed Case – Streamlines originating from within the wake of the first row

Figure 7.30 Baseline Case - Static Pressure contours over the bottom surface

Figure 7.31 Vortex Generator Installed Case - Static Pressure contours over the bottom surface

Figure 7.32 Baseline Case - Static Pressure contours over the top surface

Figure 7.33 Vortex Generator Installed Case - Static Pressure contours over the top surface

Figure 7.34 Baseline Case - Heat transfer coefficient contours over the bottom surface

Figure 7.35 Vortex Generator Installed Case - Heat transfer coefficient contours over the bottom surface

Figure 7.36 Baseline Case - Heat transfer coefficient contours over the top surface

Figure 7.37 Vortex Generator Installed Case - Heat transfer coefficient contours over the top surface

Figure 7.38 Conduction loss variation with voltage change

Figure 7.39 Percentage of conduction loss compared to the total electrical power

Figure 7.40 Variation of electrical power with adiabatic wall temperature correction

Figure 7.41 Schematic of the experimental setup and measurement zone (figure not to scale)

Figure 7.42 Plain Channel, Temperature Distribution, 21 V, 12 m/s

Figure 7.43 Baseline Case, Temperature Distribution, 21 V, 12 m/s

Figure 7.44 Vortex Generator Installed Case, Temperature Distribution, 21 V, 12 m/s
Figure 7.45 Plain Channel, Heat Transfer Coefficient Distribution, 21 V, 12 m/s

Figure 7.46 Baseline Case, Heat Transfer Coefficient Distribution, 21 V, 12 m/s

Figure 7.47 Vortex Generator Installed Case, Heat Transfer Coefficient Distribution, 21 V, 12 m/s

Figure 7.48 Six lines along which the heat transfer distribution is investigated

Figure 7.49 Percentage change of $h/h_0$ on line E

Figure 7.50 Percentage change of $h/h_0$ on line D

Figure 7.51 Percentage change of $h/h_0$ on line C

Figure 7.52 Percentage change of $h/h_0$ on line B

Figure 7.53 Percentage change of $h/h_0$ on line A

Figure 7.54 Percentage change of $h/h_0$ on line F

Figure 7.55 Streamlines seeded along the top edge of the vortex generator on the left

Figure 7.56 Streamlines seeded from a rake close to the junction of the vortex generator inner surface and lower wall surface

Figure 7.57 Closer look at Figure 7.56

Figure 7.58 Heat transfer contours along the bottom surface for the baseline and vortex generator installed cases, for CFD results obtained at $Re = 61000$ (12 m/s freestream velocity)

Figure 7.59 Heat transfer contours along the measurement surface for the baseline and vortex generator installed cases, for experimental results obtained at $Re = 61000$ (12 m/s freestream velocity)
LIST OF TABLES

Table 4.1 Heat transfer improvement when all the heat transfer surfaces - bottom, top, side walls and cylinders - are considered 107

Table 4.2 Change in the Fanning friction factor 107

Table 4.3 Heat transfer improvement when only the bottom surface in the vicinity of the vortex generator is considered 107

Table 4.4 Heat transfer and total pressure change when all the heat transfer surfaces - bottom, top, side walls and cylinders - are considered 112

Table 4.5 Heat transfer improvement when only the bottom surface in the vicinity of the vortex generator is considered 112

Table 7.1 Heat Transfer Results, Re = 17000 169

Table 7.2 Heat Transfer Results, Re = 30000 169

Table 7.3 Heat Transfer Results, Re = 61000 170

Table 7.4 Heat Transfer Results, Re = 83000 170

Table 7.5 Effect of the cylinders and cylinders plus VG combination on the overall heat transfer 185
LIST OF VIDEOS

Video 6.1 Upstream 3D Streamlines Baseline 135
Video 6.2 Upstream 3D Streamlines Fence 135
Video 6.3 Wake 3D Streamlines Baseline 136
Video 6.4 Wake 3D Streamlines Fence 136
Video 6.5 Baseline Total Pressure Contours 0.5 mm above hub 136
Video 6.6 Fence Total Pressure Contours 0.5 mm above hub 136
Video 6.7 Baseline Total Pressure Contours 3 mm above hub 136
Video 6.8 Fence Total Pressure Contours 3 mm above hub 136
Video 6.9 Baseline Total Pressure Contours 6 mm above hub 136
Video 6.10 Fence Total Pressure Contours 6 mm above hub 136
Video 6.11 Baseline Total Pressure Contours 9 mm above hub 137
Video 6.12 Fence Total Pressure Contours 9 mm above hub 137
Video 6.13 Baseline k Contours 0.5 mm above hub 138
Video 6.14 Fence k Contours 0.5 mm above hub 138
Video 6.15 Baseline k Contours 3 mm above hub 138
Video 6.16 Fence k Contours 3 mm above hub 138
Video 6.17 Baseline k Contours 6 mm above hub 138
Video 6.18 Fence k Contours 6 mm above hub 138
Video 6.19 Baseline k Contours 9 mm above hub 138
Video 6.20 Fence k Contours 9 mm above hub 138
Video 6.21 Baseline h contours over the hub 140
Video 6.22 Fence h contours over the hub 140
NOMENCLATURE

\( Re_D = \) Reynolds number based on the cylinder diameter \( = \frac{\rho U_x D}{\mu} \)

\( \rho = \) Density (kg/m\(^3\))

\( \mu = \) Viscosity (kg/m.s)

\( U_x = U_{Inlet} = \) Freestream velocity in the streamwise direction (m/s)

\( U_{profile} = \) Inlet velocity profile (m/s)

\( k = \) Turbulent kinetic energy (m\(^2\)/s\(^2\)) \( = \frac{3}{2} (U_{profile} I)^2 \)

\( \omega = \) Magnitude of vorticity (1/s) \( = \frac{k^{3/2}}{I c_\mu^{1/4}} \)

\( I_t = \) Turbulence intensity (%)

\( l = \) Turbulence length scale (m)

\( c_\mu = \) A coefficient related to the k- \( \omega \) SST turbulence model

\( h = \) Heat transfer coefficient (W/m\(^2\).K)

\( x = \) Streamwise direction

\( y = \) Lateral direction

\( z = \) Vertical direction

\( P_S = \) Static pressure (Pa)

\( P_0 = \) Total pressure (Pa)

\( C_{p_S} = \frac{P_{SCylinder} - P_{StInlet}}{\frac{1}{2} \rho U_{Inlet}^2} = \) Static pressure coefficient

\( C_P = \frac{(P_0)_{YZ plane} - (P_0)_{Inlet}}{\frac{1}{2} \rho (U_{Inlet})^2} = \) Mass averaged total pressure based \( C_p \) within complete y-z planes
\[ C_{p\text{Strip}} = \frac{(P_0)_{2 \text{ mm Strip}} - (P_0)_{2 \text{ mm Inlet Strip}}}{\frac{1}{2} \rho (U_{2 \text{ mm Inlet Strip}})^2} \] = Mass averaged total pressure based \( C_p \) within 2 mm high rectangular \( y-z \) planes

\( D = \) Cylinder diameter (m)

\( St = \) Strouhal number

\( f = \) Frequency (Hz)

\( \delta = \) Boundary layer thickness (m)

\[ j = \frac{Nu}{Re \cdot Pr^{1/3}} = \text{Colburn factor} \]

\( Pr = \) Prandtl number

\[ Nu_H = \frac{h_{m-2H}}{\lambda} = \text{Nusselt number based on channel height} \]

\[ Re_H = \frac{u_{in} \cdot 2H}{\nu} = \text{Reynolds number based on channel height} \]

\[ f = \frac{2H}{4X} \left\{ \frac{\Delta P}{\rho u_{in}^2} - K \right\} = \text{Fanning friction factor} \]

\[ K = 0.6 \sigma_{\text{channel}}^2 - 2.4 \sigma_{\text{channel}} + 1.8 = \text{Empirical expansion-contraction coefficient} \]

\( \sigma_{\text{channel}} = \frac{NW_{ch}}{W} = \text{Channel size factor} \]

\( N = \) Number of channel areas

\( W_{\text{pass}} = \) Passage width (m)

\( W = \) Domain width (m)

\[ Nu_D = \frac{h_{D}}{k_{\text{air}}} = \text{Nusselt number based on diameter} \]

\[ f_{\text{Factor}} = \frac{P_{\text{in}} - P_{\text{exit}}}{\frac{1}{2} \rho U_\infty^2} = \text{Static pressure loss factor} \]

\( V_h = \) Voltage across the heater strip (V)

\( R_h = \) Resistance of the heater strip (Ohms)
\( A_h = \text{Area of the heater strip (m}^2) \)

\( I = \frac{V_h}{R_h} = \text{The current passing through the heater (Amperes)} \)

\( T_\infty = \text{Freestream static temperature (C or K)} \)

\( T_{wall} = \text{Surface temperature on the kapton heater (C or K)} \)

\( k_{wall} = \text{Thermal conductivity of depron (W/m.K)} \)

\( T_{C1} = \text{Temperature read from the thermocouple between the depron and heater surface (C or K)} \)

\( T_{C2} = \text{Temperature read from the thermocouple between the depron and plexiglass surface (C or K)} \)

\[ q''_{\text{cond}} = k_{wall} \frac{T_{C1} - T_{C2}}{\ell_{wall}} = \text{Conductive heat loss (W/m}^2\) \]

\[ q''_{\text{total}} = \frac{V_h^2}{R_h A_h} = \text{Total heat flux (W/m}^2\) \]
ACKNOWLEDGEMENTS

I offer my deepest gratitude to my advisor, Dr. Cengiz Camci for helping and guiding me throughout my graduate school studies which have provided me a wonderful learning experience. Also, I am sincerely grateful for the assistance and invaluable contributions of my committee members Dr. Schmitz, Dr. Yavuzkurt and Dr. Miller. I also offer my thanks to Tyler Kelley for his help during the construction of my experimental setup.

I am infinitely indebted to my parents Drs. Çiğdem Gökçe, M.D and Özcan Gökçe, M.D for their unending love and all the support that they have given me my whole life. Without them, I would not be able to reach my goals and become who I am now.

I would also like to thank all my family and friends for their continuous support in my endeavors. Their presence was a key factor in helping me to continue the tradition of higher education for the third generation in my family.

And last but not least, I would like to thank all the students I had the opportunity to teach at Penn State. I discovered how much I like teaching and helping my colleagues thanks to them.
Chapter 1

Introduction

The air-breathing jet engine is the driving force of modern day aircraft. While every one of its components has unique functions and purposes, the gas turbine stands out as one of the most crucial of these components. The energy which is extracted from the fuel-air mixture by the turbine keeps the engine running, therefore ensuring the continuous flight of the aircraft. Thus, the proper and efficient operation of the gas turbine is of utmost importance.

There are many subtopics of gas turbine technology that are of unique significance; this thesis work will focus on two specific ones: The cooling flow within gas turbine blades and the flow near the nozzle guide vanes (NGV).

The total temperature at the inlet of the turbine directly contributes to two important factors: The efficiency of the turbine [1] and the amount of thrust generated by the engine [2]. In other words, the higher the value of the total temperature, the higher will the efficiency and thrust become. Consequently, it is of paramount importance to render the engine operable at the highest possible total temperature. One method of doing this will be to generate/find new materials strong enough to withstand the total temperature as well as the mechanical stress. A less complicated method of attaining higher operational total temperatures would be to improve the cooling of the blades themselves. In addition, even if stronger materials were found, cooling would still create additional performance. Therefore, improving the cooling of turbine blades will be extremely useful under any circumstance.

Increasing the heat transfer within turbine blade cooling channels by promoting turbulent mixing is a very effective way of improving the cooling. Cylindrical pin fins of various
alignments and configurations serve this purpose very well, because they create a flow of increased turbulence and unsteady character compared to the plain channel which does not contain pin fins. The flow of increased turbulent nature in turn promotes additional convective heat transfer.

**Figure 1.1** Internal layout of the gas turbine cooling channel, pin-fin cooling region circled in red [3]
Pin-fins of different shapes and types, circular and non-circular alike have been investigated in great detail. An experimental study which places rows of straight or inclined pin fins to determine their effect on heat transfer levels has been done by VanFossen [4]. This study also makes a comparison of long versus short pin fins (shorter than 4 cylinder diameters in length), leading to the heat transfer coefficients on the pin surface being 35% higher than the endwall surface, and the short pin fins creating an important increase in heat transfer levels when compared to the channel with no fins installed. Metzger and Haley [5] performed an experimental study where the effect of rows of short pin fins made out of heat conducting and non-heat conducting material had on heat transfer was analyzed. It was observed that both the non-conducting and conducting pin fins generated a similar result, and both showed an increase in heat transfer when compared to the channel with no fins installed. Uzol and Camci [6] have conducted an experimental study consisting of three different pin fin arrays. The techniques used were Liquid Crystal Thermography and Particle Image Velocimetry (PIV), for pin fin geometries which are circular, simple elliptic (SEF) and N-type (consisting of a NACA 4-digit profile). The SEF and N-type fin arrays performed better in terms of total pressure loss, whereas the circular pin fin array resulted in a 27% increased heat transfer rate compared to the other geometries. Another experimental study which investigates the effect of placing a circular pin fin with no tip clearance (distance between the fin tip and endwall) and fins with varying amounts of clearance shows that improvements in total pressure loss levels are obtained with moderate heat transfer decreases [7].

Another method to improve heat transfer is to use vortex generators (VGs) as turbulence promoters [8-10]. The specific name "vortex generator" comes from the fact that vortex generators induce additional longitudinal vortices into the domain they are inserted in. Aside from their usage in boundary layer separation control on the lifting and control surfaces of high-performance swept wings, VGs with various geometric properties are installed on flat plate fins and on plate
fins which have a pin fin (such as a circular or elliptical pin fin), in order to enhance heat transfer for certain heat exchanger and cooling applications.

In the case of a delta wing type VG, when flow approaches the VG inclined at a certain angle of attack with a finite amount of velocity, a region of high pressure is formed below the wing [8]. Conversely, a region of low pressure is formed above the wing. Friction causes the flow to separate over the side edges of the wing, and the flow begins to roll-up from below the wing towards the top. Consequently, this roll-up is shed hundreds of wing chord lengths downstream of the wing. Vortex generators also have different forms which operate on similar vortex generation principles, such as the delta winglet type vortex generator. Depending on how they are oriented with respect to the oncoming flow, the delta winglet type vortex generators are classified under two main groups: "Common flow up" and "Common flow down" [9]. When the common flow within the two legs of the longitudinal vortex is directed upwards and away from the wall, these are called common flow up type VGs. Conversely, when the common flow within the two legs of the longitudinal vortex is directed downwards and towards the wall, these are called common flow down type VGs. Common flow up leads to the thickening of the boundary layer inside the vortex wake, whereas common flow down leads to the thinning of the boundary layer within the wake. The shed vortices are called longitudinal vortices, and their axes are parallel to the surface the wing is attached to. The core of the longitudinal vortex is a high velocity, low pressure zone. Also, when a vortex generator or multiple VGs are installed on a plate fin which has a pin fin, they usually increase the size of the wake which leads to an increased pressure drop downstream of the pin fin. If they are installed on flat plate fins, then they also induce an additional pressure drop. However, this adverse effect is considered acceptable since they have a very important benefit: Due to their orientation and separation inducing thin geometric structure, the VGs cause a massive increase in turbulent mixing inside the flow. As the flow passes over sharp edge of the vortex generator, it separates and detaches from the surface. It then continues downstream in
between the confined zone created by the vortex generators in a vortical pattern. This vortical flow structure induces increased turbulent mixing into the domain, which combines the high momentum flow in the freestream with the relatively lower momentum flow near the endwall and pin-fin surfaces. Thus, the exchange of hot and cold air is promoted in the boundary layer region as well as the majority of the domain, leading to increased heat transfer over the endwall and pin-fin surfaces. This fact is put to use in many heat exchanger applications such as the cooling devices for automobile engines, which employ very shallow (width to height ratio 30-100) channels with low Reynolds numbers (around 500-2000). However, it has not been used widely for cooling purposes in gas turbine cooling channel flow. We think that investigating the benefits of the vortex generator concept will be invaluable to our current investigations which involve turbine cooling passages, as well as other disciplines which require improved heat transfer using turbomachinery scale Reynolds numbers. For gas turbine cooling channel flow, the width to height ratio varies between 1 and 8, and the Reynolds number range is between 10000 and 100000. It should be noted that, due to the increased flow blockage they add into the flow zone, the heat transfer improvements obtained using the vortex generators involve an increase in the pressure loss across the channel. Therefore, it is aimed to obtain heat transfer improvements while keeping the pressure loss increase as low as possible.

The gas turbine's efficiency is adversely affected by the viscous loss generated due to the horseshoe vortices which form around turbine NGVs and rotor blades [11]. Compared to the freestream flow, the endwall boundary layer contains a momentum deficit, which is the main reason behind the generation of the horseshoe vortex [12]. A portion of the hot gas mixture of air and fuel is transported by the horseshoe vortices from the combustor exit towards the NGVs. The aerodynamic and heat transfer characteristics of the flow around turbine inlet flow could be improved if the location at which such vortices form were known. As a result, the allowed operational total temperature at the turbine inlet would be larger, which would also imply the
presence of a higher turbine efficiency. In addition, the convection of horseshoe vortices into turbine passages which contain high speed flows creates a noticeable total pressure loss. All of these factors urge the necessity to understand and improve the conditions imposed by the horseshoe vortex.

Two physical features combine to create a roll-up within the streamwise plane of symmetry: First, the presence of the wall boundary layer leads to the separation of the fluid from the hub endwall at a wall saddle point \[13\]. Compared to the boundary layer flow the freestream has larger momentum. Due to this difference in momentum levels, the flow which stagnates near the saddle point undergoes a rolling motion \[13\]. Second, the bluff body located downstream imposes an adverse pressure gradient, which is impossible for the fluid to compensate for after a certain point \[14\]. Subsequently, the roll-up in mention progresses downstream in the shape of a horseshoe.

Various experimental studies have been conducted to investigate the behavior of horseshoe vortices and the effects that accompany them \[13\-17\]. The time dependent characteristics of and the heat transfer in the vicinity of a horseshoe vortex around a body having a symmetric airfoil profile have been studied by Praisner and Smith \[16\],\[17\]. Hada et al. observed that the heat transfer near the leading edge-hub endwall junction was strongly influenced by the leading edge diameter in their study which contains a computational validation of their experiments \[18\]. Detailed experimental analysis of horseshoe vortex formation around a vertical cylinder in cross flow has been carried out by Eckerle and Langston \[13\] and Eckerle \[14\]. Eckerle and Awad have provided a correlation based on the Reynolds number calculated using the diameter of the cylinder \[15\]. A number of researchers have shown that utilizing endwall fences placed within turbine passages and the turbine environment in general will be beneficial \[19\-21\]. The literature on horseshoe vortices and turbine aerodynamics in general suggests that modifying the endwall shape at aerodynamically crucial locations could result in fewer losses.
Building an experimental setup to investigate fluid properties inside the cooling channel and/or around the NGV blade is possible; however, it will be an expensive endeavor, both in terms of time and funds. On the other hand, thanks to the high computing capabilities of modern computers, modeling the flow within such domains with the use of Computational Fluid Dynamics (CFD) is a fast and economical method. Furthermore, the best results obtained using CFD will suggest optimum configurations, and a lot of time and resources may be saved by experimenting only with such optimum configurations.

Keeping in mind the points mentioned about turbine cooling channel flow and the flow around a NGV, and after having studied the previously published work, we conducted 3 CFD based studies \cite{22}, \cite{23}, \cite{24}. Throughout these studies, along with the circular cylinder shaped pin fin, we modeled the NGV blade as a circular cylinder, where the diameter of the cylinder is proportional to the leading edge diameter of the blade. The leading edge diameter of the NGV is the dominant factor which affects horseshoe vortex formation around the blade and near the hub, and as such, it is reasonable to use this proportionality in the numerical computations. The main idea behind these studies was to insert an endwall fence upstream of the cylinder. In the case of the flow around a NGV, the fence is expected to partially or totally relieve the adverse effects of the horseshoe vortex from the flow domain, and successfully modify the location of the main roll-up and subsequent horseshoe vortex. These goals are desired to be achieved without affecting the freestream region. The results indicated that these goals were attained \cite{22}, \cite{23}, \cite{24}. For the flow around a NGV, the rectangular endwall fence successfully modified the shape and location of the main horseshoe vortex roll-up \cite{22}, \cite{23}. The endwall fence will yield decreased interaction between consecutive horseshoe vortices if used in a turbine NGV row \cite{22}, \cite{23}. Furthermore, when placed at an even bigger distance upstream of the cylinder, the endwall fence will result in slight spanwise total pressure as well as local, mass averaged total pressure improvements downstream of the cylinder \cite{24}. 
During our studies involving the endwall fence and the flow around the NGV, we observed that the endwall fence significantly increases the turbulent mixing inside the domain. Turbulent mixing is a great contributor to improving heat transfer, therefore we decided to investigate the potential effect of using the endwall fence inside the turbine cooling channel flow as well [24]. The presence of the endwall generates an increase in turbulent mixing and turbulent kinetic energy levels within the cylinder wake. Hence, a higher amount of heat transfer - an average of 10% increase when compared to the baseline case with no fence installed - towards the hub surface is observed within the cylinder wake [24]. This is an important result for circular pin fins, because increased heat transfer coefficient levels over the hub surface implies that improved cooling can be achieved through the application of such fences.

The final part of this study focuses on the turbine cooling channel flow. Inspired by the results we obtained regarding heat transfer, we decided to investigate the effect of the vortex generator on heat transfer in our domain. We inserted two pairs of common flow up oriented, delta winglet type vortex generators inside a channel which contains an array consisting of three rows of staggered cylinders, and investigated the flow behavior at various Reynolds numbers. The computational study of this setup resulted in an important finding: The vortex generators increase the exchange of high and low momentum flow inside the domain, which results in significantly improved heat transfer, as high as a 27% overall increase compared to the baseline case which has no vortex generators involved. This improvement was accompanied by a 13% increase in static pressure loss across the channel, which is an acceptable amount for gas turbine cooling channel flow. We also carried out an experimental study to verify our computational results: Experiments conducted in the open loop wind tunnel located in the Turbomachinery Aero-Heat Transfer Lab show that the vortex generators result in an overall 14% heat transfer increase compared to the baseline case, over the constant heat flux surface, validating the general
trends obtained during the CFD study. The static pressure loss increase across the channel associated with the experiment was 8%.

The CFD simulations were carried out with the commercially available software FLUENT which is developed by ANSYS. The meshing software used, GAMBIT, is also a product of ANSYS. The k-ω model with Shear Stress Transport (SST) has been used due to its reputation for generating good results with flows containing adverse pressure gradients. The Reynolds numbers we have used in our simulations range between 11000 and 83000.

Our work has the unique characteristic of applying an endwall shape modification previously employed by non-aerospace industries for cooling applications, to the gas turbine cooling channel flow which is vital to aerospace propulsion for the first time.

The conclusions drawn from this study are very important because they demonstrate that the usage of vortex generators generate a noticeable improvement in the cooling properties of the cooling channel flow. As a result, the operational performance of gas turbines will be improved. In addition to the aerospace industry, the improved heat transfer properties will also be beneficial for any discipline and application which contains channel flow using a similar Reynolds number range and channel width to height ratio. An example to these additional fields would be cooling applications for computers.
Chapter 2

Literature Review: Past Studies focusing on Heat Transfer and Performance Increase in Channel Flow and Turbine Flow

In order to obtain information which is crucial to our analysis of the turbine channel cooling flow and the flow development around the NGV blade, an important number of studies previously published have been investigated. This chapter provides the main aspects of some of these studies beginning with horseshoe vortex formation, continuing with endwall shape modification and concluding with pin-fins and cooling applications.

Eckerle’s PhD thesis contains a detailed study of horseshoe vortex formation around a cylinder in cross flow [14]. The results show the formation of a single, main vortex in the plane of symmetry, and no other vortices were present. The formation of the horseshoe vortex and whether multiple vortices form or not depend on the Reynolds number used. The measurements of flow conditions within the plane of symmetry showed good agreement with the potential flow solution. Eckerle and Langston conducted an experimental investigation of the horseshoe vortex formation around a cylinder in cross flow [13]. They have carried out extensive measurements of pressure and velocity, in addition to obtaining visual images of the surface flow. The horseshoe vortex was completely within the endwall boundary layer, and its extending legs tended to move away from the main flow axis. Eckerle and Awad found a new parameter which depended on the Reynolds number based on the cylinder diameter and the momentum thickness [15]. This number correlated well with their own experimental results and those of two other published studies.

Goldstein and Karni investigated the effect of the endwall boundary layer and three-dimensional secondary flows on heat and mass transfer [25]. They have noted that due to the interaction between the endwall boundary layer and the cylinder, the resultant three-dimensional
secondary flows increased the heat transfer within the cylinder wake at distances up to 3.5 cylinder diameters from the wall. Praisner and Smith analyzed the temporal dynamics of the horseshoe vortex and the resulting heat transfer around a symmetric airfoil shaped vertical body\textsuperscript{[16],[17]}. Four different vortex formations within the symmetry plane have been observed, and the secondary vortex which forms right below the saddle point is replaced by a fluid inrush, which implies the displacement of the endwall boundary layer from the wall and its subsequent mixture with the horseshoe vortex. Similar to these studies in content, Hada et al. conducted an experimental work on flow around a symmetric airfoil\textsuperscript{[18]}. It is concluded that the leading edge diameter has an important influence on the endwall heat transfer near the endwall-leading edge junction. In addition, they conclude that the endwall heat transfer coefficient is directly proportional to the Reynolds number based on the leading edge diameter of the airfoil.

Modifying the shape of the endwall to improve flow attributes within turbines has been the subject of a number of studies\textsuperscript{[19-21]}. Most of these studies focused on the results of placing endwall fences within turbine passages to control cross flow. The potential benefits of placing a boundary layer fence on the endwall to reduce the effects of passage vortices on turbine passage flow has been investigated by Camci and Rizzo\textsuperscript{[19]}. They found that, within a 90 degree turning duct which had a square cross section of 203 mm by 203 mm, all cases involving a fence height of 12.7 mm resulted in decreased total pressure loss when compared to the no-fence case, which shows that endwall fences mounted on the endwall have promising characteristics which can improve turbine performance. Rizzo's experimental work shows that a thin fence which extends from the $0^\circ$ to $90^\circ$ planes, has a height equal to half the boundary layer thickness, has a width of 4.7 mm and is located in the middle of the pressure and suction sides of the duct reduces secondary flow effects by modifying the shape of the passage vortex and results in a reduction of the passage averaged total pressure loss by 6.5\%, when compared to the no-fence case\textsuperscript{[20]}. Prümper tested a great number of configurations to analyze the effect of placing boundary layer
fences on turbine endwalls, and found that cascade losses are reduced and the application of fences could especially be beneficial to cascades having short blades \cite{21}. Turgut's thesis contains a detailed analysis of nonaxisymmetric endwall contouring and leading edge modifications on turbine NGV flow \cite{26}. His study showed that leading edge fillets successfully minimized horseshoe vortex formation in front of the blade and that endwall contouring reduced the cross flows going from the pressure side to the suction side of the vane. CFD results indicate a 7% reduction in total pressure loss, while the experimental results indicated a 15% reduction.

The idea of using protrusions installed on the endwall or lower surface of a flow to improve heat transfer has been the subject of many previous studies. These protrusions are usually named pin-fins and they have various shapes and profiles, one of the most common geometries being the circular cylinder. VanFossen's experimental study investigates the effect of inserting rows of straight or inclined circular pin-fins on channel flow heat transfer \cite{4}. It was found that short pin-fins created an important increase in heat transfer levels when compared to the plain channel with no pin-fins inserted. An experimental study where the potential benefits of using rows of short pin fins made out of heat conducting and non-heat conducting material showed that, both the non-heat conducting and heat conducting pin-fins generated a similar result\cite{5}. Furthermore, an increase in heat transfer when compared to the plain channel was observed in both cases. In addition to working with circular pin-fins, Uzol and Camci have analyzed the effects of using the simple elliptic (SEF) and N-type (having a NACA 4-digit profile) geometries for pin-fins \cite{6}. Liquid Crystal Thermography and Particle Image Velocimetry (PIV) methods indicate that an array consisting of circular pin-fins resulted in a 27% improvement in heat transfer rate when compared to the SEF and N-type pin-fin arrays, while the SEF and N-type pin-fin arrays generated a lower total pressure loss across the channel due to decreased flow blockage. In their experimental study, Chang et al. observed that a circular pin-fin with no tip clearance (distance between the pin-fin tip and endwall) and pin-fins with varying
amounts of clearance created reductions in total pressure loss with moderate heat transfer decreases\textsuperscript{7}.

The concept of using vortex generators as heat transfer promoters has been researched by a number of authors \textsuperscript{8-10}, \textsuperscript{27-32}. Pauley and Eaton's experimental work investigates the effects of placing delta winglet type VGs on a flat plate fin, and they compare the results of the cases with VGs to the baseline case which contains only the plate fin \textsuperscript{10}. The researchers analyze the influence of various parameters on heat transfer performance downstream of the two pairs of common flow down type VGs and notice that all spacing (lateral separation between the VGs) values generate noticeable heat transfer enhancement within the longitudinal vortex wake. Russell et al.'s study contains an analysis of various vortex generator configurations placed on the channel endwall \textsuperscript{27}. They conclude that rectangular winglets are the most effective heat transfer promoters, especially when placed close to each other in a staggered array formation. An experimental work which investigates the effect of placing delta wing, delta winglet pair, rectangular wing and rectangular winglet pair type of vortex generators on a plate fin, and compares the results with the baseline case containing no VGs was published by Fiebig et al. \textsuperscript{8}. They found that the delta winglet pair of VGs generates heat transfer enhancements of up to 60\% percent over the fin surface. Also, delta winglet pairs and delta wings are found to be superior to their rectangular counterparts in general. Chen and co-workers' numerical investigation contains a detailed analysis of various combinations of VGs and pin-fins of elliptical nature \textsuperscript{28}. Throughout the numerous studies they have conducted, the authors concluded that the combination of the elliptical tube and common flow down type delta winglet pairs is the optimum way to enhance heat transfer. They have obtained heat transfer improvements ranging from 50\% to 87\%, but they also mention that this increase is accompanied by a noticeable increase in total pressure loss. Torii and co-workers' experimental study is one of the few which uses common flow up type delta winglet pairs with staggered or in-line arrays of circular tubes \textsuperscript{29}. They have found that the
common flow up type delta winglet pairs placed in the vicinity of a staggered array of circular tubes, not only shows heat transfer enhancement, but also manages to achieve this by decreasing the pressure loss by 40% at higher Reynolds numbers.

The literature review contained in this chapter provided additional support to our initial thoughts about the potential benefits of using VGs as heat transfer enhancers in gas turbine cooling channel flow, and using endwall fences in improving the flow in the vicinity of the NGV as well as the gas turbine cooling channel flow.
Chapter 3

Numerical Analysis to Obtain Increased Heat Transfer and Performance in Channel Flow and Turbine Flow

This chapter contains the details of the numerical (computational) portion of our research. It begins with the description of the theoretical background of the turbulence model used, in addition to the main aspects of the software utilized. It also includes the geometrical layout and mesh structure of the investigated cases. The numerical modeling of the flow was conducted in FLUENT, version 14, a commercially available software developed by ANSYS. The generation of the flow domain and the meshing of the grid was done in GAMBIT, another software developed by ANSYS.

3.1 Theoretical Background: The k-ω SST Turbulence Model

Two-equation turbulence models are widely used in numerical studies due to their ability to serve as a fine compromise between accuracy and fast computing. The first researcher to work on a two-equation turbulence model was Kolmogorov \(^{[33]}\). He chose to use the turbulent kinetic energy, \(k\), and the dissipation rate per unit turbulence kinetic energy, \(\varepsilon\), as the two turbulence parameters to be used in the model. He modeled the two differential equations which determine the variation of these two parameters, and created the model. His dissipation rate transport equation was based on Helmholtz's equation written for the magnitude of vorticity, \(\omega\).
Wilcox [34] later continues along Kolmogorov's thought process, by starting with the turbulence kinetic energy. The turbulence kinetic energy, or the kinetic energy per unit mass of the turbulent fluctuations, \( k \), can be calculated as [35]:

\[
k = \frac{1}{2} \bar{u}' \bar{u}' = \frac{1}{2} (\bar{u}^2 + \bar{v}^2 + \bar{w}^2)
\] (3.1)

The eddy viscosity can be expressed as [34]:

\[
\mu_T = \text{constant} \cdot (\rho \sqrt{k} l)
\] (3.2)

where \( l \) is the turbulent length scale, and \( u', v', w' \) are the fluctuation velocity components. The Reynolds-stress tensor is given by [34]:

\[
\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3} \rho k \delta_{ij}
\] (3.3)

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\] (3.4)

where \( S_{ij} \) is the mean strain rate tensor. The final form of the turbulence kinetic energy equation is written as:

\[
\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \rho \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]
\] (3.5)

Based on Equation 3.3, Kolmogorov made the reasonable assumption of considering \( k \) to be proportional to \( \nu_T \) (dynamic viscosity), since the dimensions of \( \nu_T \) are (\text{length})^2/(\text{time}) and those of \( k \) are (\text{length})^2/(\text{time})^2. As a result, the dimensions of \( \nu_T/k \) are (time). The dimensions of the turbulence dissipation parameter \( \varepsilon \) are (\text{length})^2/(\text{time})^3. Thus, \( \varepsilon/k \) has dimensions 1/(\text{time}). The statement of an equation in terms of \( \omega \) is enabled by having a variable
which has units of \(1/(time)\). Using the vorticity transport concept, Kolmogorov obtained Equation 3.6 after a dimensional analysis and results obtained from observing the physical nature of the problem:

\[
\rho \frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = -\beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ \sigma \mu_T \frac{\partial \omega}{\partial x_j} \right]
\]  

(3.6)

The k-\(\omega\) model is one of the most commonly used two-equation turbulence models in numerical computations. FLUENT contains Wilcox’s k-\(\omega\) model \(^{[34]}\) and Menter’s k-\(\omega\) with SST model \(^{[36]}\) as part of its turbulence models.

Wilcox’s model utilizes the following essential equations and parameters:

**Kinematic Eddy Viscosity:**

\[
\mu_T = \frac{\rho k}{\omega} \quad (3.7)
\]

**Turbulence Kinetic Energy:**

\[
\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma^* \mu_T) \frac{\partial k}{\partial x_j} \right]
\]  

(3.8)

**Specific Dissipation Rate:**

\[
\rho \frac{\partial \omega}{\partial t} + \rho U_j \frac{\partial \omega}{\partial x_j} = \alpha \omega \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma \mu_T) \frac{\partial \omega}{\partial x_j} \right]
\]  

(3.9)

**Closure Coefficients and Auxiliary Relations:**

\[
\alpha = \frac{5}{9}, \quad \beta = \frac{3}{40}, \quad \beta^* = \frac{9}{100}, \quad \sigma = 0.5, \quad \sigma^* = 0.5
\]

(3.10)

\[
\epsilon = \beta^* \omega k, \quad l = \frac{k^{0.5}}{\omega}
\]

(3.11)
The model is shown to be accurate for a wide variety of turbulent flow problems, and it is recommended that the interested reader should consult Reference 34 for additional information.

The k-ω model with Shear Stress Transport is essentially a modification of Wilcox's original k-ω model. Menter developed it and it accounts for a common basic problem of two equation models, which is their inability to foresee the onset and extent of separation when a strong adverse pressure gradient is present[36].

The transport of the principal turbulent shear stress is better represented by the eddy viscosity definition of the SST model:

\[ \nu_t = \frac{a_1 k}{\max(a_1 \omega; \Omega F_2)} \] (3.12)

\( \Omega \) is the absolute value of the vorticity, and \( F_2 = \tanh(\arg_2^2) \), where

\[ \arg_2 = \max(2 \frac{\sqrt{k}}{0.99 \omega y}; \frac{500v}{y^2 \omega}) \] (3.13)

Additional detail related to the development of this model can be found in Menter's original publication[36].

The cylinders, and when installed, the endwall fences and vortex generators create a strong adverse pressure gradient. In addition, the SST k-ω model provides a fine balance of accuracy and speed while conducting the numerical computations. These two factors have led us to decide on using the SST k-ω model for our calculations.

### 3.2 Details of the Computations and Flow Conditions

This subsection describes which settings were used in FLUENT during the CFD calculations, in addition to the flow and boundary conditions. Before moving on to the specifics
of the fence installed cases and vortex generator installed cases, the common points of both will be explained.

The solver used in our computations is a pressure based and implicit solver. The solution control parameters are the unknowns in the Navier-Stokes equations, namely the x, y, z velocity, the turbulence kinetic energy $k$, the magnitude of vorticity $\omega$, continuity and energy terms. The solutions were run until a steady convergence for each of these parameters was attained. The convergence criterion for all these parameters was to have residual levels of $10^{-3}$ or less, where 6 out of 7 of the residual levels were on the order of $10^{-5}$ or less. Second order discretizations were used for all the differential terms in the turbulence equations contained in the solution scheme. The pressure-velocity coupling was set to the coupled mode. The k-$\omega$ model with SST turbulence model was utilized. The flow was assumed to be fully turbulent, since it accurately represents both the cooling channel flow and the flow around the NGV. The flow is incompressible.

The inlet static pressure, which is also the reference static pressure for the entire domain, is the standard atmospheric pressure of 101325 Pa. The air density is 1.225 kg/m$^3$, and the viscosity of air is $1.7894 \times 10^{-5}$ kg/m.s.

### 3.2.1 Fence Installed Cases

The inlet of the domain is a velocity inlet. The side surfaces and the top surface are assigned a symmetry boundary condition. The lower surface, the cylinder, and when present the fence, are all set to the wall boundary condition. The outlet of the domain is an outflow type boundary condition. This boundary configuration constitutes a good way of representing the flow around a NGV, since the symmetry type boundary condition is useful in representing the nature of this type of flow. It is also a useful setup for the cooling channel flow.
The flow inside the domain of the fence installed case has a Reynolds number of 11000. At this Reynolds number region, the wake of the singular cylinder exhibits a periodic oscillation. In order to accurately capture all flow patterns, the simulations for the cases containing the rectangular endwall fence were carried out using a time-dependent solver.

The inlet velocity profile of this study contains a boundary layer, and its velocity values are provided by extensive measurements conducted during a previously published study at our Axial Flow Turbine Research Facility (AFTRF)\[37\].

The turbulence criteria were the turbulent kinetic energy $k$ and the magnitude of vorticity $\omega$. They depend on the velocity profile and are calculated at each data point as follows:

\[
\begin{align*}
\frac{1}{2}k & = \frac{3}{2} (U_{\text{Profile}} l t)^2 \quad (3.14) \\
\omega & = k^{1/2} \div l C_\mu^{1/4} \quad (3.15)
\end{align*}
\]

where $l_t$ is the freestream turbulence intensity, $U_{\text{Profile}}$ is the inlet velocity profile, $l$ is the turbulence length scale and $C_\mu$ is a constant related to the k-$\omega$ SST turbulence model. The freestream turbulence intensity is set to 1%, because this value is an accurate representation of the conditions of the flows being analyzed. The turbulence length scale is chosen as 18 mm, due to the fact that it is equal to the cylinder height, which represents the maximum possible eddy length. $C_\mu$ is equal to 0.09.

The time step for the unsteady calculations is selected on the experimentally proven Strouhal number of 0.2 for a circular cylinder\[38\]. Since the cylinder diameter is equal to 12 mm and the freestream velocity is 14 m/s, the natural frequency and natural time step of the oscillations is found by the following equations:

\[
f = \frac{U_\infty \cdot St}{D} \quad (3.16)
\]
Using Equation 3.16, the natural time step is found to be equal to 4.3 ms (0.0043 s). In order to accurately distinguish flow variations at each time step, the actual time step used in the calculations is \(1/5^{\text{th}}\) of the natural value. A total of 200 time steps are completed to make sure that the time variations of the flow were accurately resolved. In addition, each time step contains 20 pseudo time steps for reaching steady state convergence within that time step.

The heat transfer aspect of the flow is captured by including the energy equation in the flow solution. The temperature difference between the gas and the walls is set to 200 K. This temperature difference between the gas and the walls enables a large enough heat transfer to occur for visualization and comparison.

### 3.2.2 Vortex Generator Installed Cases

The inlet of the domain is a velocity inlet. The side surfaces, the top and bottom surfaces, the staggered rows of cylinders, and when present the vortex generators, are all set to the wall boundary condition. The outlet of the domain is an outflow type boundary condition. In real life, since the cooling channel flow occurs within a region completely surrounded by solid surfaces, the best way to analyze this type of flow with CFD is by setting the two sides, the top and the bottom of the domain to the wall boundary condition.

The flow inside the domain of the vortex generator installed cases has a Reynolds number varying between 17000 and 83000. Since we are interested in observing the time-averaged overall heat transfer change, we ran these simulations in steady mode.

The inlet velocity profile of this study contains a boundary layer, and its velocity values are found by using the \(1/7^{\text{th}}\) power law:
The freestream velocity for the cases vary between 3.39 m/s and 16.5 m/s. The boundary layer thickness is set to 1 mm, both on the top and bottom surfaces of the domain. Since this is a good real-life representation of the velocity profile located inside the open loop wind tunnel of the Turbomachinery Aero-Heat Transfer Laboratory, it is also utilized in our CFD modeling.

The turbulence criteria were the turbulence length scale and the freestream turbulence intensity. The turbulence length scale is set to 2 mm, and the freestream turbulence intensity is set to 2% \[^37\].

The heat transfer aspect of the flow is captured by including the energy equation in the flow solution. The temperature of the gas is set to 298 K and the temperature of the walls is set to 348K. The net temperature difference between the gas and the walls enable a large enough heat transfer to occur for visualization and comparison.

### 3.3 Grid Independence Study

One of the most important parts of a CFD based analysis is the grid independence study. As the name implies, the goal of the study is to make sure that the results of the computations are independent of the mesh size. In other words, the results of the CFD runs should be very close to each other, at and above a certain mesh size. The mesh size is equal to the total number of three dimensional, hexahedral cells that are contained in the structured mesh.

The procedure of the grid independence study begins by selecting a reasonable mesh size for initial calculations. Deciding whether a mesh size is reasonable or not depends entirely on the application, and it requires a certain amount of experience and know-how with CFD software. During our previously published studies, we determined that a mesh size on the order of 1 million
three dimensional hexahedral cells generated accurate results \cite{22, 23}. Since our current work is involved with a bigger domain, as well as new geometric modifications, after numerous runs we decided that a mesh size of approximately 3 million three dimensional hexahedral cells will be enough to generate accurate results.

One of the most effective ways of determining grid independence is to assign it to a significant parameter. The static pressure coefficient is indeed such a parameter, and is calculated as follows:

\[
C_{PS} = \frac{P_{Cylinder} - P_{Inlet}}{\frac{x}{2} \rho U_{Inlet}^2}
\]  \hspace{1cm} (3.19)

where the static pressure on the cylinder surface is calculated along the circumference at the middle of the cylinder’s height, and the static pressure at the inlet is the mean static pressure at the inlet of the domain. We used the middle cylinder of the first row of staggered cylinders located in our VG installed cases.

\textbf{Figure 3.1} The static pressure coefficient calculated along the mid-height circumference of the middle cylinder of the first row
Figure 3.1 shows that our results are indeed independent of the size of the structured grid. Since all three mesh sizes generate very close results, we decided that a mesh size of 3 million is ideal for our cases involving the vortex generator. For the cases employing the rectangular fence, this number varied between 1 million and 3 million.

### 3.4 Numerical Steps of Attaining Performance Increase using Fence Related Effects

The details of the flow domain generation for the fence installed cases and the subsequent meshing will be explained in this subsection.

#### 3.4.1 Geometrical Setup: Baseline Case

The computational domain is a rectangular field which has a height of 18 mm, a width of 63 mm and a length of 336.9 mm. A vertical circular cylinder of 12 mm diameter and 18 mm height is placed inside the field, and the origin of the coordinate system is placed at the center of the lower circular surface of the cylinder.
Since one of the goals of the study is to analyze the flow around a NGV, the domain and the cylinder are created based on the dimensions of a NGV previously used in our lab during the studies of Turgut [26], Kavurmacioglu et al. [39, 40] and Rao et al. [41]. The mid-span axial chord length of the NGV blade mentioned which equaled 112.3 mm, is chosen as a length scale; the domain covers the area between a point one mid-span axial chord length upstream of the cylinder center, and a point two mid-span axial chord lengths downstream. The cylinder diameter is twice the size of the leading edge diameter of the NGV blade. Previous studies show that the leading edge diameter of a NGV blade is the dominant factor in creating horseshoe vortices [15]. Therefore, we simplified our analysis and modeled the NGV blade as a circular cylinder. The width of the domain is 63 mm and 5.25 times the cylinder diameter, and the height of the domain is 18 mm and equal to the cylinder height. All domain dimensions were chosen as such because they are sufficiently large to accurately capture and model all flow phenomena. Furthermore, even though the domain is created based on the dimensions of a NGV blade, it can also very
effectively symbolize a circular pin fin. The height-to-diameter ratio of the cylinder is 1.5, which is a reasonable value for pin fins which are considered as heat transfer promoters \cite{4,5}.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure3.3.png}
\caption{Certain dimensions of the baseline case, all of which also apply to the fence installed case.}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure3.4.png}
\caption{Side view of the baseline case demonstrated in the x-z plane.}
\end{figure}
3.4.2 Mesh Structure: Baseline Case

The flows which we are interested in contain large gradients at various locations, especially in the boundary layer and near the junction of the endwall and cylinder. In order to capture all the details of the flow and present an accurate representation of the conditions inside the domain, we have preferred to use a structured grid in our computations.

Due to the complexity of the flow inside the boundary layer and in the vicinity of the cylinder, the mesh inside and near these zones contain a high amount of cells. The resulting fine grid successfully shows the high amount of variations within these areas.

As we move towards the top of the domain, we place nodes with increasing distance. Since the flow in the freestream is uniform, decreasing the cell density within that zone is a reasonable way of saving computational time. Furthermore, the same principle is applied to the mesh as we move from the cylinder leading edge towards the inlet, and as we move from the
cylinder trailing edge towards the outlet. Since the flow gradually becomes fully developed as we move away from the cylinder, the decreased cell density does not affect the accuracy of the solution.

**Figure 3.6** The gradually decreasing mesh density near the top of and at the inlet of the domain

**Figure 3.7** The gradually decreasing mesh density as we move towards the outlet
3.4.3 Geometrical Setup: Fence Installed Case

The domain of the fence-installed case is the same as the baseline case except for the addition of a rectangular endwall fence. A rectangular fence is placed 24 mm upstream of the cylinder center in order to successfully alter the aerodynamic and thermal properties of the flow. This specific location was chosen due to its potential to achieve the goals we mentioned in previous chapters: The distance between the fence and the cylinder will serve to generate an increased roll-up, which will in turn generate an increase in heat transfer rates downstream of the cylinder, as well as an increase in total pressure levels downstream of the cylinder. The heat transfer enhancement will serve the cooling channel flow, while the rise in total pressure levels downstream of the cylinder will help the flow around the NGV.

Figure 3.8 Three dimensional overview of the fence installed case
Figure 3.9 Certain dimensions of the fence-installed case

Figure 3.10 Cross section details of the fence
**Figure 3.11** Side view of the fence-installed case demonstrated in the x-z plane

**Figure 3.12** Side views of each case demonstrated together for comparison
3.4.4 Mesh Structure: Fence Installed Case

The mesh structure of the fence installed case is very similar to the baseline case. The crucial aspects which were considered when creating the baseline case remain valid for the fence installed case as well. The boundary layer region and the vicinity of the cylinder is densely meshed, and the locations near the inlet and outlet contain a lower density of three dimensional cells.

The vital addition to the mesh of the fence installed case is near the fence itself. Since high gradients for various flow parameters are present due to the addition of the fence, the regions close to all four sides and the top of the fence need to be filled with a high number of cells in order to appropriately resolve these gradients. Figures 3.14 and 3.15 demonstrate how the presence of the fence affects the mesh structure: The tightly spaced nodes near the mesh translate all the way back towards the inlet, in order to create a structured mesh.

Figure 3.13 The dense mesh structure near the fence, cylinder and in the boundary layer
Figure 3.14 The gradually decreasing mesh density near the top of and at the inlet of the domain

Figure 3.15 The gradually decreasing mesh density as we move towards the outlet
3.5 Numerical Steps of Attaining Heat Transfer Increase using Vortex Generator Induced Effects

Inspired by the results of our validation study which is described in detail in Chapter 4, we applied the concept of placing common flow-up oriented delta winglet type vortex generator pairs in a channel containing three rows of staggered cylinders, onto a channel design of our own. The basic principles of the concept are implemented onto a test section geometry which is congruent to the Reynolds number range suitable for gas turbine cooling channel flow. The range is between 15000 and 100000, therefore all our numerical simulations are carried out accordingly.

At the end of the validation study, our idea of using a half height vortex generator supported our expectations of giving better results, therefore we decided to utilize it in our actual CFD runs and experiments.

3.5.1 Geometrical Setup: Baseline Case

The width and height of the entire domain exactly match the dimensions of our open loop wind tunnel test section which is described in detail in Chapter 5. The length of the computational domain is 40 cm longer than the actual test section and is equal to 167 cm, in order to make sure the flow has room to fully develop inside the far downstream wake during the CFD run. The positioning of the staggered array of cylinders and the diameter (1/5 of the duct width), is a combination of our reasoning and Torii et al.’s [29] methodology. We have chosen to leave a considerable amount of channel space between the cylinders in order to place vortex generators which would be large enough to create the flow effects we desired.
Figure 3.16 Overview of the baseline case’s domain

Figure 3.17 Top view detailing the positions and dimensions of the staggered cylinder array
3.5.2 Mesh Structure: Baseline Case

Figures 3.18 through 3.21 demonstrate certain portions of the baseline case’s mesh. The meshing strategy utilized is the same as we have used before: The locations close to the wall surfaces have densely spaced nodes to ensure the proper resolution of gradients at those locations, and the regions close to the freestream have fewer cells to ensure a reasonable total number of cells. The baseline case has a total of 2.8 million hexahedral structured cells.

Figure 3.18 Overview of the baseline case’s mesh
Figure 3.19 A volume near the cylinder depicting the mesh structure

Figure 3.20 Another volume near the cylinder depicting the mesh structure
3.5.3 Geometrical Setup: Half Height Vortex Generator Installed Case

The domain of the half height vortex generator installed case is the same as the baseline case’s domain, except for the addition of the two pairs of common flow up oriented vortex generators. The positioning and sizing of the vortex generators was done using a combination of our own reasoning and the general layout proposed by Torii et al. [29]. Since we are interested in turbine cooling channel flow, the aspect ratio (width / height) of our domain’s cross section is much smaller, compared to Torii et al.’s setup [29]. Furthermore, we have paid attention to keeping the ratio of the cylinder height to the cylinder diameter very close to 1, due to the fact that this is a widely preferred configuration for gas turbine cooling channel flow.
Figure 3.22 Overview of the half height vortex generator installed case’s domain

Figure 3.23 Top view of the half height vortex generator installed case’s domain
Figure 3.24 Side view of the half height vortex generator installed case’s domain
3.5.4 Mesh Structure: Half Height Vortex Generator Installed Case

The mesh of the half height vortex generator installed case follows our main meshing strategy of keeping zones close to wall surfaces of any type (cylinders, side, top and bottom walls, vortex generators) densely meshed.

Figure 3.25 Overview of the baseline case’s mesh
Figures 3.26 and 3.27 denote an important aspect of the mesh: The mid-height of the domain contains closely spaced nodes similar to the top and bottom endwall surfaces. This is due to the necessity of having similar edge node distributions on various volumes near the vortex generators. The structure contained inside these volumes is transported all the way to the inlet in order to form a complete structured grid.

Figure 3.28 shows that the structure of the half height vortex generator installed case’s grid is directly related to the half height vortex generator shape: The triangular faces directly above the vortex generators are split into 4 quadrilateral zones. This enables the construction of a fine, well performing mesh.
Figure 3.27 Another look at the volumes in the vicinity of the vortex generators and cylinders

Figure 3.28 Volumes directly above the vortex generators
Chapter 4
Numerical Validation of the Vortex Generator Concept

Even though the vortex generator concept has been utilized in various fields, especially in heat exchanger applications which fall under the scope of mechanical engineering, it has not been used for the purpose of improving the heat transfer characteristics of the gas turbine blade cooling channel. Therefore, to our knowledge, our study is the first to analyze the effect of inserting vortex generators into the gas turbine blade cooling channel, and to investigate the results of using them in combination with circular pin-fins inside the blade cooling channel.

4.1 General Information on Vortex Generators

The specific name "vortex generator" comes from the fact that vortex generators induce additional longitudinal vortices into the domain they are inserted in. VGs are used for boundary layer separation control on the lifting and control surfaces of high-performance swept wings. Flat plate fins and plate fins which have a protrusion (such as a circular or elliptical pin-fin) have VGs mounted on them in order to enhance heat transfer levels in heat exchangers.
Figure 4.1 shows a delta wing type vortex generator as an example to the vortex generator configuration: A region of high pressure is formed below the wing when flow with a finite velocity approaches the wing which is inclined at a certain angle of attack. On the other hand, a region of low pressure is formed above the wing. The flow then separates over the side edges of the wing due to friction, and it begins to roll-up from below the wing towards the top. As a result, this roll-up is shed hundreds of wing chord lengths downstream of the wing.

Vortex generators may also have different forms which operate on similar vortex generation principles, such as:
Delta winglet type vortex generators are classified under two main groups based on their orientation with respect to the oncoming flow: "Common flow up" and "Common flow down". When the common flow within the two legs of the longitudinal vortex is downwards and towards the wall, these are called common flow down type VGs. On the other hand, when the common flow within the two legs of the longitudinal vortex is upwards and away from the wall, these are called common flow up type VGs. Common flow down leads to the thinning of the boundary layer inside the vortex wake, whereas common flow up leads to the thickening of the boundary layer within the wake.

Figure 4.2 Common VG geometries [8]
Figure 4.3 Common flow up $^{[9], [42]}$

Figure 4.4 Common flow down $^{[9], [42]}$
The shed vortices are called longitudinal vortices, and their axes are parallel to the surface to which the wing is attached. A high velocity, low pressure zone is located inside the core of the longitudinal vortex. Also, when a vortex generator or multiple VGs are installed on a fin which has a protrusion, they usually increase the size of the wake which leads to an increased pressure drop downstream of the protrusion. They also induce a pressure drop in the event they are installed on plain fins. However, this adverse effect is considered acceptable since they have a very important benefit: The longitudinal vortices shed downstream increase the exchange of hot and cold air, between the boundary layer region and the freestream region. In other words, they act as heat transfer enhancers. This feature is taken advantage of in many heat exchanger applications.

After reviewing a large number of published studies on vortex generators and how they are used in heat transfer enhancement applications [8-10]. [27-32], we decided that Torii et al.’s [29] method of using common flow up oriented delta winglet type vortex pairs would be the most effective way to achieve our goals of attaining heat transfer improvements in the gas turbine cooling channel.

In their experimental work, Torii et al. place two pairs of common flow up oriented delta winglet type vortex pairs into the gaps of the first row of cylinders, which is part of a staggered array of eight cylinders distributed into three rows.
Figure 4.5 Dimensional information and layout of Torii et al.’s setup

Torii et al.’s experimental facility consisted of a small open loop wind tunnel which contained a rectangular test section. The air flow was provided by a 1.5 kW blower, and the heating of the experimentation surfaces was done via a stainless steel ribbon. Temperature and pressure measurements were carried out using sensors and pressure probes, respectively.

They utilized the Colburn and Fanning factors as their performance criteria. The Colburn factor is a measure of heat transfer levels, and is defined as:

$$ j = \frac{Nu}{Re Pr^{1/3}} \quad (4.1) $$

where $Nu$ is the Nusselt number, $Re$ is the Reynolds number and $Pr$ is the Prandtl number. The Nusselt number and Reynolds number are defined as follows:
\[ Nu_H = \frac{h_m.2H}{\lambda} \]  \hspace{1cm} (4.2)

\[ Re_H = \frac{U_{in}.2H}{\nu} \]  \hspace{1cm} (4.3)

where \( h_m \) is the average heat transfer coefficient over their measurement area, \( H \) is the test section height, \( \lambda \) is the thermal conductivity of air, \( \nu \) is the viscosity of air and \( U_{in} \) is the inlet freestream velocity.

The Fanning friction factor \( f \) is a quantity which stands for an averaged static pressure loss coefficient. It is defined as:

\[ f = \frac{2H}{4X} \left\{ \frac{\Delta P}{\rho U_{in}^2} - (K_c + K_e) \right\} \]  \hspace{1cm} (4.4)

where \( K_c \) and \( K_e \) are the expansion and contraction coefficients, respectively. These coefficients are an estimate of the static pressure loss generated by the friction over the surfaces of the wind tunnel. \( \Delta P \) is the difference in static pressure between the exit and the inlet of the domain, and \( X \) is the axial location at which \( f \) is being calculated.
Figure 4.6 Torii et al.'s heat transfer and total pressure change data for the staggered array of cylinders \cite{29}

Figure 4.6 denotes the ratio of the Colburn factor of the vortex generator installed case to that of the baseline case, and the ratio of the Fanning friction factor of the vortex generator installed case to that of the baseline case. As the figure indicates, it is possible to obtain a 10% increase in heat transfer with a pressure loss reduction of 40%, at certain Reynolds numbers. The combination of both of these beneficial effects is most likely the result of the low Reynolds numbers used in conjunction with high channel width to height ratio.
4.2 Numerical Validation

The findings of Torii et al.’s study \cite{29} prompted us to study the concept even further. We decided to re-model their experimental domain via CFD, and wanted to get a deeper understanding of the underlying physics behind the vortex generator concept.

4.2.1 Numerical Method

Even though the numerical methodology is similar to the process described in Chapter 3.2, some details are worth mentioning. The inlet of the domain is a velocity inlet. The side surfaces, the top and bottom surfaces, the staggered rows of cylinders, and when present the vortex generators, are all set to the wall boundary condition. The outlet of the domain is an outflow type boundary condition.

The CFD runs were conducted for flow Reynolds numbers of 2000 and 500. The simulations do not include unsteadiness, since we are interested in the overall performance of the vortex generators instead of instantaneous effects. The inlet velocity is chosen to be uniform and equal to 9.8 m/s.

The turbulence criteria were the turbulence length scale and the freestream turbulence intensity. The turbulence length scale is set to 0.5 mm, and the freestream turbulence intensity is 5%.

The heat transfer aspect of the flow is captured by including the energy equation in the flow solution. The temperature of the gas is set to 298 K and the temperature of the walls is set to 348 K.
4.2.2 Geometrical Setup

Our re-modeling of their domain is very similar to what Torii et al. had constructed in their experimental setup. The width, length and height of the domain, the dimensions of the cylinders and vortex generators, as well as their relative position with respect to each other all match Torii et al.’s study. However, we made some slight modifications to enable the creation of a structured mesh. We made the height of the vortex generator and the height of the domain equal to each other, whereas the authors had preferred to have a small gap between the top of the VG and the top of the domain.

![Figure 4.7 Geometrical layout of the baseline case (figure not to scale in the z-direction for enabling easier visualization)](image-url)
Figure 4.8 Geometrical layout of the baseline case seen from the top

Figure 4.9 Geometrical layout of the vortex generator installed case (figure not to scale in the z-direction for enabling easier visualization)
Figure 4.10 Certain details of the vortex generators seen from the top
Figure 4.11 Overview of the baseline and vortex generator installed cases seen together for comparison (figure not to scale in the z-direction for enabling easier visualization)
4.2.3 Mesh Structure

The meshing strategy required the diligent creation of sub-volumes in order to allow a structured mesh. The regions near the side walls, as well as the top and bottom surfaces contain a high density mesh. The vicinity of the vortex generators and the cylinders are also filled with a high number of cells in order to successfully capture all flow phenomena.

Figure 4.12 Overview of the baseline case’s mesh (figure not to scale in the z-direction for enabling easier visualization)

Figure 4.13 Overview of the vortex generator installed case’s mesh (figure not to scale in the z-direction for enabling easier visualization)
Figure 4.14 through 4.17 show the mesh development process as well as certain vital grid locations which were carefully created for the baseline case.

**Figure 4.14** The volumes which form the baseline case’s geometry

**Figure 4.15** A face mesh which demonstrates the densely placed boundary layer nodes near the upper and lower surfaces
Figure 4.16 The placement of the nodes near to and away from the cylinder surfaces

Figure 4.17 Closer look at the hexahedral cells inside one particular volume
Figures 4.18 through 4.27 depict the detailed mesh development process for the vortex generator installed case. The general meshing strategy is the same as the one used for the baseline case. However, the locations near and upstream of the vortex generators contain a high number of meticulously created small volumes. Each of these volumes contain a group of three-dimensional cells which come together to form a well designed, complete structured grid.

**Figure 4.18** The volumes which form the vortex generator installed case’s geometry
Figure 4.19 Top view of the vortex generator installed case

Figure 4.20 Closer look at the inlet of the domain
**Figure 4.21** Closer look at the region encircled in red in Figure 4.20

**Figure 4.22** Closer look at the volumes in the vicinity of the vortex generators
Figure 4.23 The volumes downstream of the vortex generators which are very similar to their baseline counterparts

Figure 4.24 Certain volumes which are highlighted to demonstrate the mesh structure
Figure 4.25 Closer look at a specific volume which lies in between a pair of vortex generators

Figure 4.26 A volume adjacent to a cylinder
Figure 4.27 The densely spaced nodes near the top and bottom surfaces will provide good flow resolution
4.3 The Half Height Vortex Generator Concept

In addition to focusing on understanding and validating the concept proposed by Torii et al., we came up with an idea of our own during this process. We wanted to see the results of using a vortex generator which is half as high as the domain. Since the half height vortex generator creates a smaller amount of flow blockage inside the channel, we expected to see a heat transfer enhancement similar to that created by the full height vortex generator, but with a smaller amount of pressure loss across the channel.

The numerical method used for the baseline and vortex generator installed cases is also utilized for the half height vortex generator installed case. Figures 4.28 through 4.31 demonstrate the geometrical layout and certain details of the mesh structure belonging to the half height vortex generator installed case.

![Figure 4.28 Geometrical layout of the half height vortex generator installed case (all dimensions are the same as the full height vortex generator installed case, except for the height of the half height VG)]
Figure 4.29 Overview of the half height vortex generator installed case’s mesh

Figure 4.30 Close look at the mesh structure near one of the half height vortex generators
Figure 4.31 One of the half height vortex generators and some of the volumes near the VG highlighted

Figure 4.31 shows the essence of the uniqueness of the half height vortex generator installed case’s grid: The space above the half height vortex generator is divided into smaller volumes so that each sub-volume has a structured form which is in accord with the entire structured grid.
4.4 Re = 2000 Results

4.4.1 Two and Three Dimensional Streamlines

Figure 4.32 Streamlines in the streamwise plane of symmetry for the baseline case

Figure 4.33 Streamlines in the streamwise plane of symmetry for the full height vortex generator installed case
When Figures 4.32 and 4.33 are compared, it is observed that the roll up patterns upstream of the cylinder (central cylinder in the first row) are the same for both cases. However, the wake structures behave differently; the baseline case exhibits no vortical roll-up at the cylinder trailing edge, but the vortex generator has induced a vortical roll-up at the same cylinder's trailing edge.
Figure 4.34 contains the comparison of all three cases, the baseline, full VG and half VG respectively:

Figure 4.34 Comparison of the two dimensional streamlines for all cases
The roll-up patterns for both vortex generator installed cases are very similar. They both differ from the baseline case in the sense that their wakes have a more turbulent structure. This is verified by the three dimensional streamlines.

**Figure 4.35** Three dimensional streamlines of the baseline case

**Figure 4.36** Three dimensional streamlines of the full height vortex generator installed case
When Figures 4.35 and 4.36 are compared, the effect of the vortex generators is easily observed: The streamlines which separate over the edges of the vortex generators move downstream in a vortical pattern. This flow behavior induces increased turbulent mixing between the high momentum containing free-stream flow and the relatively lower momentum boundary layer flow. Furthermore, the momentum exchange is also increased in regions close to the cylinder surfaces and side walls. These effects combine to yield increased heat transfer inside the domain.

Figure 4.37 includes the three dimensional streamlines for the half height vortex generator installed case as well: The wake of the half height vortex generator installed case behaves similar to the full height vortex generator installed case. However, it directs more of the flow towards the center of the channel compared to the full height VG.
Figure 4.37 Comparison of the three dimensional streamlines for all cases
4.4.2 Static Pressure Contours over the Bottom Surface

Figures 4.38 and 4.39 show the static pressure contours over the bottom surface:

**Figure 4.38** Static Pressure contours over the bottom surface for the baseline case

**Figure 4.39** Static Pressure contours over the bottom surface for the full height vortex generator installed case
Figures 4.38 and 4.39 demonstrate that the increased blockage generated by the vortex generators results in a lower static pressure in the wake downstream of the last row of cylinders.

Figure 4.40 demonstrates the benefit of the half height vortex generator: The pressure levels in the far downstream wake of the cylinders are higher compared to the full vortex generator installed case. This is supported by the improved Fanning friction factor values which will be given in Chapter 4.4.4.
Figure 4.40 Comparison of the static pressure contours along the bottom surface: Baseline, full VG and half VG, respectively
4.4.3 Heat Transfer Coefficient Contours over the Bottom and Top Surfaces

**Figure 4.41** Heat transfer coefficient contours for the baseline case - bottom surface

**Figure 4.42** Heat transfer coefficient contours for the full height vortex generator installed case - bottom surface
When Figures 4.41 and 4.42 are compared, the immediate wakes of the vortex generator pairs exhibit a noticeable heat transfer increase. It is also noticed that the flow recovers and behaves mostly like the baseline case once it reaches the second row of cylinders. The increased heat transfer values in the vicinity of the vortex generators are a result of the impingement of the vortical structure generated by the vortex generators on the endwall surface.
Figures 4.43 and 4.44 demonstrate that the addition of VG pairs improves the heat transfer characteristics over the top surface as well:

**Figure 4.43** Heat transfer coefficient contours for the baseline case - top surface

**Figure 4.44** Heat transfer coefficient contours for the full height vortex generator installed case - top surface
Figure 4.45 Heat transfer coefficient contours for the half height vortex generator installed case - bottom surface

Figure 4.46 Heat transfer coefficient contours for the half height vortex generator installed case - top surface
Figure 4.47 Comparison of heat transfer contours over the bottom surface of each case: Baseline, full VG and half VG, respectively.
Figure 4.47 demonstrates the effects of the full height VG and the half height VG: Both VG configurations generate a noticeable overall heat transfer increase. Figures 4.41 through 4.47 support the results of Chapter 4.4.1: Compared to the baseline case, the bottom and top surfaces of the vortex generator installed cases contain increased heat transfer coefficient contours. This means that the increased exchange of momentum suggested by the streamline patterns is indeed taking place inside the entire domain, resulting in improved heat transfer levels compared to the baseline case with no vortex generators installed.

4.4.4 Heat Transfer and Pressure Change Results

The shaded area in Figure 4.48 is the surface on which the mean heat transfer coefficient is calculated (when only the bottom surface is considered) for all cases, and the line denoted by the blue arrow is used to calculate the static pressure change within the domain.

Figure 4.48 Static pressure change calculation line and part of the heat transfer calculation regions
The following tables demonstrate the heat transfer changes obtained by using the vortex generators, when calculated over various regions:

**Table 4.1** Heat transfer improvement when all the heat transfer surfaces - bottom, top, side walls and cylinders - are considered

<table>
<thead>
<tr>
<th></th>
<th>Full VG</th>
<th>Half VG</th>
</tr>
</thead>
<tbody>
<tr>
<td>(j/j_{\text{baseline}})</td>
<td>1.084</td>
<td>1.120</td>
</tr>
</tbody>
</table>

**Table 4.2** Change in the Fanning friction factor

<table>
<thead>
<tr>
<th></th>
<th>Full VG</th>
<th>Half VG</th>
</tr>
</thead>
<tbody>
<tr>
<td>(f/f_{\text{baseline}})</td>
<td>1.060</td>
<td>0.985</td>
</tr>
</tbody>
</table>

**Table 4.3** Heat transfer improvement when only the bottom surface in the vicinity of the vortex generator is considered

where:

\[
\frac{j}{j_{\text{baseline}}} = \frac{Nu}{Re.Pr^{1/3}} 
\]  

\[
f = \frac{2H}{4X} \left\{ \frac{\Delta P}{\rho \bar{U}_{\text{in}}^2} - K \right\} 
\]  

\[
K = 0.6 \sigma_{\text{channel}}^2 - 2.4 \sigma_{\text{channel}} + 1.8
\]  

\[
\sigma_{\text{channel}} = \frac{NW_{\text{pass}}}{W}
\]  

\[N = \text{Number of channel areas (2 - the regions between the cylinders)}\]

\[W_{\text{pass}} = \text{Passage width (45 mm)}\]

\[W = \text{Domain width (150 mm)}\]
Tables 4.1 through 4.3 verify the conclusions which we drew from the heat transfer coefficient contours, and serve as another proof that the vortex generators are beneficial in improving heat transfer characteristics. The full height vortex generator creates a 8% increase in overall heat transfer when compared to the baseline case, but at the price of a Fanning friction factor increased by 6%. The half height vortex generator creates an even higher overall heat transfer change, 12%, with the addition of a decrease in the Fanning friction factor by 1%. This is a very important result. Furthermore, as demonstrated by Table 4.3, the vortex generators are especially effective in enhancing the heat transfer in their immediate vicinity: The full height VG shows a 19% increase, and the half height VG shows a 17% increase. The improvements in heat transfer are the resulted of increased momentum exchange inside the domain promoted by the vortex generators, and the increase in static pressure loss is a result of the extra flow blockage generated by the vortex generators.
4.5 Re = 500 Results

Since Torii et al. investigated the effects of using the vortex generator with low Reynolds numbers (200-2000), we also wanted to analyze the effects of using the vortex generator at a Reynolds number of 500. The turbulence model was set to laminar for these runs.

These runs were conducted for the baseline and full height vortex generator cases only. Since we are primarily interested in heat transfer improvements, pressure results are not given here for conciseness.

4.5.1 Two and Three Dimensional Streamlines

![Comparison of the two dimensional streamlines](image.png)

**Figure 4.49** Comparison of the two dimensional streamlines
Figure 4.49 shows that both upstream and downstream of the cylinder, the behavior of the roll-up pattern is similar for both cases. It should also be noted that the wake roll-ups seen for the Re = 2000 case is not seen in the laminar case.

Figure 4.50 Comparison of the three dimensional streamlines

Figure 4.50 shows that the laminar flow behaves as expected for the baseline case: The streamlines follow the curvatures generated by the cylinders. The VG pairs introduce turbulence into the domain, therefore the wake structures for these two cases are quite different.
Nevertheless, the combination of low Reynolds number flow and the VG pairs does not seem to be as efficient as its turbulent counterpart.

4.5.2 Heat Transfer Coefficient Contours over the Bottom Surface

![Image of heat transfer coefficient contours]

**Figure 4.51** Comparison of heat transfer coefficient contours over the bottom surface

Figure 4.51 demonstrates that even though there is a positive change in heat transfer levels near the VG pairs, the laminar flow structure fails to generate an overall useful change in heat transfer.
4.5.3 Heat Transfer and Pressure Change Results

<table>
<thead>
<tr>
<th>$j/j_{\text{baseline}}$</th>
<th>0.985</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f/f_{\text{baseline}}$</td>
<td>1.048</td>
</tr>
</tbody>
</table>

Table 4.4 Heat transfer and total pressure change when all the heat transfer surfaces - bottom, top, side walls and cylinders - are considered

| $j/j_{\text{baseline}}$ | 1.097 |

Table 4.5 Heat transfer improvement when only the bottom surface in the vicinity of the vortex generator is considered

For a Reynolds number of 500, the overall heat transfer characteristics are negatively affected with the addition of vortex generator pairs. However, there is a 10% increase heat transfer levels in the vicinity of the vortex generators. This comes at the price of an increase in the Fanning friction factor value by 5%.

4.6 Conclusions of the Validation Study

At the end of our validation study which involved analyzing the setup proposed by Torii et al. [29], we noticed that the vortex generator concept is indeed very effective in improving heat transfer. We are primarily looking to enhance heat transfer in gas turbine cooling channel flow with an acceptable pressure loss, therefore we were satisfied with the improvements shown during our validation study. Since the half height vortex generator gives heat transfer improvements similar to the full height vortex generator with a lower amount of pressure loss, the half height vortex generator is used in our CFD runs and experiments.

Later, we created our own setup involving the vortex generator concept. Its details are given in Chapter 3.5, and its results are given in Chapter 7.
Chapter 5

Experimental Procedure for the Validation of Numerically Obtained Heat Transfer Improvements

In order to verify the numerical results we have obtained, we constructed an experimental setup and made detailed heat transfer measurements for a Reynolds number belonging to one of our cases. We used the open loop wind tunnel located in the Turbomachinery Aero-Heat Transfer Laboratory.

5.1 Experimental Setup

The open loop wind tunnel which was used is made up of the following components, respectively: An axial air blower, a plenum chamber with multiple screens to ensure the uniformity of the flow, a circular nozzle with a high area ratio, a transition duct which goes from a circular profile to a rectangular one, a converging nozzle, the test section, a diverging nozzle and a diffuser which discharges the air into the ambient. An infrared camera is placed a short distance away from the test section in order to capture the temperature distribution along the test section endwall surface. The open loop wind tunnel and the infrared camera combine to form our experimental setup. Figures 5.1 and 5.2 depict the external view and schematic of our setup, respectively.
The axial air blower which is used to direct ambient air towards the test section uses a 7.5 kW electric motor to operate a 45.7 cm diameter fan, and can generate a pressure change of 15 cm of water for various flow rates. The air speed inside the wind tunnel can be changed between 0 and 30 m/s via an adjustable frequency AC operator which controls the electric motor.

Figure 5.1 External view of the experimental facility
Once the air exits the blower, it goes through a series of screens within the plenum chamber in order to reach uniform flow. The flow then continues through a circular nozzle which has an area ratio of 8.65, which in turn directs the flow into a 137 cm long duct. This duct has a convergent profile which transitions from a circular to a rectangular profile. The rectangular cross section of the duct is 36.67 cm by 15.24 cm. The duct is followed by a rectangular converging nozzle which ends with a cross section of 36.67 cm by 7.62 cm.

The test section is situated at the end of the converging nozzle. It is a straight duct 127 cm long, and has a cross section of 36.67 cm by 7.62 cm. The length of the test section ensures that the flow has enough distance to settle down once it traverses the array of cylinders and VGs. Its width allows the insertion of an array of 4 cylinders and 4 half cylinders which captures the effect of using multiple pin-fins and VGs in conjunction. Its height allows a duct height-to-cylinder diameter ratio very close to 1, which is a well performing setup \textsuperscript{[6],[43]}.

The final components of the setup are a diverging rectangular nozzle which has an exit cross section of 36.67 cm by 15.24 cm, and a diffuser which discharges the air back into the ambient surroundings.

Figure 5.2 Schematic of the test section \textsuperscript{[43]}
Figure 5.3 External view of the plenum chamber, circular nozzle and converging duct  

Figure 5.4 Schematic of the converging duct [43]
5.2 Details of the Test Section

The test section walls are made of 1.2 cm clear acrylic. The clear acrylic (plexiglass) provides an easy visual observation of the experiment. As mentioned in the previous subsection of this chapter, the dimensions of the test section are chosen large enough to accurately capture all aspects of flows which fall into the Reynolds number range provided by the tunnel.

The array of cylinders located in the test section consists of 4 cylinders and 4 semi-cylinders to generate the staggered array formation. The first row of cylinders is located 32.5 cm downstream of the test section entrance. The VGs are located inside the passages contained between the first row of cylinders. The cylinders are 7.4 cm high and have a diameter of 7.33 cm. The VGs are 7.33 cm long and are 3.81 cm high. The cylinders and the VGs are painted with black paint to prevent radiative heat losses. The cylinders and VGs are constructed from commercially available plastic tubing. This material is chosen for its relative simplicity in
forming the pieces via a turning lathe and a cutter. The pieces are mounted on the test section using double sided tape and balsa pieces for additional support.

Figure 5.6 The cylinders, semi-cylinders and vortex generators painted in black

The test section endwall is covered with 2 mm thick depron to minimize conductive losses. The surface of the depron as well as the thermofoil heater is painted black with black paint to prevent radiative heat losses.

The heating of the measurement zone is done using a 25 cm by 37.5 cm thermofoil heater produced by Minco. The heater is supplied voltages varying from 6 V to 21 V in order to create the same amount of constant heat flux for both the baseline and VG-installed cases. The voltage is applied via a DC power supply which provides up to 24 V and draws up to 60 A.
Figure 5.7 The thermofoil heater

Figure 5.8 The test section covered with depron, including the portion below the thermofoil heater
The test section surface painted in black

The temperatures used in the conduction loss calculation across the depron are measured via two thermocouples. One is inserted in between the heater and depron surface. The other one is inserted between the depron and plexiglass surface.
Figure 5.10 Thermocouple between the depron and plexiglass
Figure 5.11 Intermediate thermocouple installation step

Figure 5.12 Thermocouple between the heater and depron
Another thermocouple for calibrating the infrared camera's emissivity value is installed on the surface of the heater.

**Figure 5.13** Final look of the conduction loss thermocouple setup

**Figure 5.14** Camera calibration thermocouple
Figure 5.15 Conduction loss thermocouple plugs

Figure 5.16 Calibration thermocouple plug and the heater power cable
The test section main window contains a special portion which contains a 30 cm by 30 cm infrared window. The portion of the test section endwall which is situated above the second row of cylinders and the end portion of the first row of cylinders is cut out, and replaced by the infrared window. This is done in order to ensure that the infrared camera is able to capture the temperature values within the area of measurement. The infrared window is produced by Edmund Optics. The infrared window is secured to the surface using epoxy and aluminum tape.

The final modifications on the plexiglass window were to mount a thermocouple probe to measure the freestream temperature, a Pitot velocity probe to measure the freestream velocity and inlet static temperature, and a static pressure port to measure the exit static pressure. The temperatures are measured using a thermometer datalogger produced by Extech. The Pitot velocity probe and the static pressure port are connected to a pressure measurement device to obtain the necessary readings.
Figure 5.18 The infrared window secured on the plexiglass

Figure 5.19 The thermocouple probe and the Pitot velocity probe
**Figure 5.20** The thermometer datalogger

**Figure 5.21** The pressure measurement device used for measuring velocity and static pressure
The T-620 designated infrared camera used to measure the temperature distribution on the endwall is produced by FLIR Systems. It is a multi-functional camera which is capable of operating within a wide range of temperatures such as -40 degree Celsius to 250 degree Celsius, and 100 degree Celsius to 600 degree Celsius. It is mounted on a tripod and positioned directly across the infrared window in order to capture the temperature distribution on the endwall surface. It is calibrated using the heater surface thermocouple reading to use the correct value of emissivity, and its transmission setting is calibrated to match the infrared window's transmission levels.

![Infrared camera](image)

**Figure 5.22** Infrared camera
Figure 5.23 The final look of the test section when the cylinders are installed

Figure 5.24 The final look of the test section when the cylinders and vortex generators are installed
Figure 5.25 Final look of the experimental setup
5.3 Details of the Experimental Procedure

The following steps are used to obtain the heat transfer coefficient values over the measurement surface:

1. The AC operator is turned on and set to the appropriate frequency for the desired tunnel speed.
2. The infrared camera is turned on and is made ready for taking infrared images when the flow conditions are attained.
3. The DC power supply is connected to the thermofoil heater and the power supply is set to the desired voltage, and the blower is turned on via the AC operator control panel.
4. 15 minutes are allowed to pass with the system on and the infrared image is taken.

Joule's first law states that the total generated heat flux on the rectangular heater strip is given by:

\[ q''_{total} = \frac{V_h^2}{R_h A_h} \]  

(5.1)

where,

- \( V_h \) = Voltage across the heater strip
- \( R_h \) = Resistance of the heater strip
- \( A_h \) = Area of the heater strip
- \( I = V_h / R_h \) = The current passing through the heater

In order to account for the conduction heat loss, the following term must be included in the convective heat transfer coefficient calculation:

\[ q''_{cond} = k_{wall} \frac{T_{c1} - T_{c2}}{t_{wall}} \]  

(5.2)
where,

\[ T_{wall} = \text{Surface temperature on the thermofoil heater} = \text{Obtained through the infrared camera} \]

\[ k_{wall} = \text{Thermal conductivity of depron} = 0.035 \text{ W/m.K}^{[44,45]} \]

\[ T_{C1} = \text{Temperature read from the thermocouple between the depron and heater surface} \]

\[ T_{C2} = \text{Temperature read from the thermocouple between the depron and plexiglass surface} \]

Finally, the convective heat transfer \( h \) is calculated as:

\[
    h = \frac{q''_{\text{total}} - q''_{\text{cond}}}{T_{\text{wall}} - T_{\infty}} \tag{5.3}
\]

where,

\[ T_{\infty} = \text{Freestream static temperature} = \text{Measured via thermocouple probe} \]

The emissivity value of the infrared camera is fixed once the heater surface thermocouple reading matched the temperature read by the infrared camera. The DC power supply is set to voltage values which are high enough to generate a noticeable amount of temperature difference between the freestream flow and the endwall.

The value of \( h \) is calculated at each temperature data point obtained from the infrared image. The data points are obtained by exporting the JPG image generated by the camera to FLIR’s Quickreport software, then transferring the temperature data obtained at each pixel to MATLAB. Then, the heat transfer coefficient at each pixel is calculated using Equation 5.3. The results for the plain channel, baseline case and the VG installed cases are then compared based on the calculated \( h \) values via contour plots drawn in MATLAB. The results for the experiments are presented in Chapter 7.4 and the experimental uncertainty analysis is presented in Appendix A.
Chapter 6

Results of the Computational Study for determining Performance Increase due to Fence Related Effects

This chapter contains the results of the setup described in Chapter 3.4, and gives a detailed discussion on how the upstream endwall fence is used to generate beneficial effects for turbine cooling channel flow and the flow around a NGV.

6.1 Two and Three Dimensional Streamlines

This section focuses on explaining the changes in flow patterns using the two-dimensional streamlines within the streamwise plane of symmetry and the three-dimensional streamlines inside the entire domain. Rakes placed at the same exact locations, which contain the same exact streamline density for both the baseline and fence-installed cases are used to create the plots and videos. The videos demonstrate the time-dependent properties of the runs. All videos can be accessed by clicking on the respective link for each video on the electronic copy of this dissertation.
Figure 6.1 Baseline case: Streamlines within the streamwise plane of symmetry

Figure 6.2 Fence-installed case: Streamlines within the streamwise plane of symmetry

The comparison of Figure 6.1 and Figure 6.2 yields that the amount of vortical roll-up within the streamwise plane of symmetry is increased due to the installation of the fence. Downstream of the cylinder and near the fence-endwall junction, turbulent kinetic energy and
heat transfer levels will rise as a result of the enhanced vortical roll-up which promotes the mixing of the high momentum fluid contained in the freestream with the low momentum fluid inside the boundary layer.

The three dimensional flow patterns upstream of the cylinder are depicted in Videos 1 and 2, for the baseline and fence-installed cases, respectively. Unsteady vortex shedding is successfully captured in both cases, indicating that the chosen time-step was accurate enough to capture the scale of the turbulence. Comparing the two cases, it is noted that when an upstream fence is employed, the legs of the horseshoe vortex rise to higher locations along the height of the domain, downstream of the cylinder. Since the increased roll-up brings more high-momentum fluid towards the endwall, the main vortex contains more energy within itself for the fence-installed case. This leads to increased turbulent mixing inside the domain.

**Video 6.1:**
**Upstream 3D Streamlines Baseline**

**Video 6.2:**
**Upstream 3D Streamlines Fence**

The structure of the flow inside the cylinder wake is visualized in Videos 3 and 4, for the baseline and fence installed cases, respectively. When compared to the wake of the baseline case, the wake of the fence-installed case contains a more complex three-dimensional pattern, indicated by the more three-dimensional vortical structure it contains. This is another indicator of increased turbulent mixing inside the domain, as a result of the fence’s presence.
6.2 Total Pressure Contours

The two and three dimensional streamline analysis pointed to increased turbulent mixing and momentum transport within the domain, when a fence is employed. To investigate what kind of effects this situation has on the total pressure distribution, contours of the parameter in mention along x-y planes placed 0.5, 3, 6 and 9 mm above the hub, respectively, are grouped and presented in the following videos for each case:

**Video 6.5:**
Baseline Total Pressure Contours 0.5 mm above hub

**Video 6.6:**
Fence Total Pressure Contours 0.5 mm above hub

**Video 6.7:**
Baseline Total Pressure Contours 3 mm above hub

**Video 6.8:**
Fence Total Pressure Contours 3 mm above hub

**Video 6.9:**
Baseline Total Pressure Contours 6 mm above hub

**Video 6.10:**
Fence Total Pressure Contours 6 mm above hub
The videos indicate that the total pressure levels at the cylinder leading edge-endwall junction 0.5 mm above the hub are higher for the fence installed case. However, the converse of this is true at the location immediately downstream of the fence. This is due to the large impinging and detaching sides of the roll-up: The large detaching side takes high-momentum fluid away from the endwall and reduces the total pressure near the fence. On the other hand, the large impinging side which is close to the cylinder leading edge-endwall junction brings high-momentum fluid towards the endwall and elevates the total pressure. A similar observation is made for the total pressure contours located 3 mm above the hub, because the increased roll-up due to the installation of the fence is still effective at this height. The horseshoe vortex effects completely vanish at 6 mm and 9 mm above the hub. Thus, the goal of not modifying the freestream flow after having installed the fence is achieved.

A signal of a possible local improvement in total pressure is seen 0.5 mm above the hub: The total pressure levels in the cylinder wake are higher for the fence-installed case. Furthermore, the unsteady vortex shedding covers a larger area downstream of the cylinder and the vortices shed encompass a larger distance along the y-axis. A thinner, smaller secondary horseshoe-vortex like structure which is created by the sharp edges of the fence is the main factor behind this phenomenon.

Two additional points worthy of mention are observed via the total pressure contours: A more complex shedding mechanism involving a highly turbulent vortical structure is present within the wake of the fence-installed case. In addition, the thin, low total pressure region
between the main horseshoe vortex and the secondary horseshoe vortex generated by the fence remarks the effect the fence has on local changes in total pressure.

6.3 Turbulent Kinetic Energy Contours

The videos below represent the turbulent kinetic energy $k$ distribution along x-y planes placed 0.5, 3, 6 and 9 mm above the hub for both cases:

Video 6.13:
Baseline $k$ Contours 0.5 mm above hub

Video 6.14:
Fence $k$ Contours 0.5 mm above hub

Video 6.15:
Baseline $k$ Contours 3 mm above hub

Video 6.16:
Fence $k$ Contours 3 mm above hub

Video 6.17:
Baseline $k$ Contours 6 mm above hub

Video 6.18:
Fence $k$ Contours 6 mm above hub

Video 6.19:
Baseline $k$ Contours 9 mm above hub

Video 6.20:
Fence $k$ Contours 9 mm above hub
The turbulent kinetic energy contour videos confirm the results obtained from the streamline analysis and total pressure contours: The high momentum fluid brought towards the endwall by the large impinging side of the main vortical roll-up creates an increase in turbulent kinetic energy levels near the cylinder leading edge-endwall junction when a fence is employed, and its large detaching side reduces these levels immediately downstream of the fence.

Similar to the observations made using the total pressure contours, the turbulent kinetic energy contours obtained at each height show that the presence of the fence results in a larger area downstream of the cylinder being covered by vortex shedding. The turbulent kinetic energy contours drawn 0.5 mm above the hub is another proof to the complex three-dimensional nature of the wake of the fence installed case.

When the fence is employed, the turbulent kinetic energy levels within the cylinder wake are higher at all heights, most specifically 0.5 mm above the hub. The increased level of turbulent kinetic energy is a direct indicator to a potential increase in heat transfer levels.
6.4 Heat Transfer Coefficient Contours over the Hub

The heat transfer coefficient contour variations over the endwall surface are shown for each case by Videos 21 and 22:

Video 6.21:
Baseline h contours over the hub

Video 6.22:
Fence h contours over the hub

When compared to the baseline case, a region of low heat transfer exists just downstream of the fence, while a large heat transfer region exists near the cylinder leading edge-endwall junction. This means that the upstream heat transfer coefficient levels are also affected by the large main vortical roll-up imposed by the fence.

The comparison of the wake region of each case exhibits a very important result: The magnitude of heat transfer coefficient levels downstream of the cylinder is higher when the fence is used. This implies that the usage of an upstream endwall fence will generate increases in heat transfer levels within the cylinder wake. This is a good result for cooling pin fin applications within the cooling channels of turbine blades because enhancements in cooling will enable higher turbine total temperatures, which will lead to better turbine performance and output.
6.5 Spanwise Total Pressure Distribution

\[ \Delta P_0 = P_{0_{\text{fence}}} - P_{0_{\text{baseline}}} \]

**Figure 6.3** The difference between total pressure values for the two cases, in the spanwise direction

Figure 6.3 shows gage total pressure values which are averaged over a series of one dimensional (along the y-axis, domain width) lines having the same streamwise coordinates but varying spanwise (z-direction, along the cylinder height) coordinates. Each z level is assigned an average total pressure value, and the process is repeated for three different locations downstream of the cylinder trailing edge. The difference in these values for the fence-installed and baseline case are plotted in Figure 6.3. The legend denotes the ratio of the streamwise (x-direction) distance between x-coordinate of each line and the cylinder trailing edge, to the diameter of the...
cylinder. The results are the average of values taken at 11 time steps, placed 0.4 ms apart. The cylinder is represented by the blue shape.

It is observed that the farther away the data is taken downstream from the cylinder trailing edge, the total pressure levels tend to decrease. Furthermore, the presence of the fence creates an increase in total pressure levels, especially for the 1D and 2.5D locations. Even though these are small local gains, this situation is still beneficial for gas turbine applications, since higher total pressure levels in the vicinity of the NGV will result in higher efficiency.
6.6 Heat Transfer Coefficient Levels

**Figure 6.4** The convective heat transfer coefficient magnitude over the hub along the streamwise axis, plotted for the two cases.

**Figure 6.5** The percentage change in $h$ when a fence is employed, when compared to the baseline case.
The convective heat transfer coefficient, \( h \), is averaged over one-dimensional (in the y-direction, hub width) lines; then, each x/D location is assigned the arithmetic mean of its corresponding line. Figure 6.4 demonstrates the actual values for each case, and Figure 6.5 represents the percentage change in the values when a fence is installed. The green square shows the location of the fence, and the red rectangle shows the location of the cylinder.

The presence of the fence increases the magnitude of the heat transfer rate towards the endwall, downstream of the cylinder. The heat transfer increase takes place in a region which extends up to 7-8 cylinder diameters downstream of the cylinder trailing edge. The average value of the enhancement mentioned is around 10% when compared to the baseline case. This situation is beneficial for circular pin fin applications in internal cooling passages, where an increased heat transfer rate will result in more effective cooling. The averaged values of the heat transfer coefficient confirm the results obtained from examining the heat transfer coefficient contours, where the videos indicated larger values of \( h \) over the endwall surface.
6.7 Mass Averaged Total Pressure Based Cp Analysis

Figure 6.6 Mass averaged total pressure based Cp values over entire y-z planes

\[ C_p = \frac{(P_0)_{YZ\ plane} - (P_0)_{Inlet}}{\frac{1}{2} \rho (U_{Inlet})^2} \]  \hspace{1cm} (6.1)

Figure 6.7 Mass averaged total pressure based Cp values over 2 mm-high rectangular planes extending above the hub surface
Figure 6.6 was plotted using Equation 6.1, and Figure 6.7 was plotted using Equation 6.2.

Figure 6.6 shows mass averaged total pressure based $C_p$ values over $y$-$z$ cross sections taken at various locations along the length of the domain. It is observed that for the entirety of the domain, $C_p$ does not vary greatly when a fence is employed. However, as Figure 6.7 demonstrates, if the first 2 mm-region above the endwall is analyzed separately, the fence creates a noticeable increase in total pressure levels downstream of the cylinder trailing edge. This is an important feature because any rise in total pressure within the turbine NGV flow is a positive change. It should be noted that due to the extra flow blockage created by the fence, a total pressure drop is observed right at the location of the fence, also indicated by the large detaching side of the main roll-up.
Chapter 7

Results of the Computational and Experimental Study for determining Heat Transfer Increase due to Vortex Generator Related Effects

This chapter describes the computational results of the configuration described in Chapter 3.5 and the experimental results of the setup given in Chapter 5. It also gives a detailed discussion on how the common flow up oriented delta winglet type vortex generators are used to generate beneficial effects for the gas turbine cooling channel flow: The chapter concludes with the detailed explanation of the physics which make the vortex generator concept useful to our aim of improved cooling.

7.1 Reynolds Number Based Computational Fluid Dynamics Analysis

In order to investigate what kind of influence the Reynolds number has on flow properties when the vortex generators are utilized, four different turbomachinery scale Reynolds numbers were chosen. These are 17000, 30000, 61000 and 83000. The following subsections include the comparison of certain important parameters and graphs.
7.1.1 Two Dimensional Streamlines

Figure 7.1 Re = 17000, Baseline and Vortex Generator installed cases, respectively
Figure 7.2 Re = 30000, Baseline and Vortex Generator installed cases, respectively
Figure 7.3 Re = 61000, Baseline and Vortex Generator installed cases, respectively
Comparing Figures 7.1 through 7.4, it is noticed that the baseline case shows similar properties for Re = 17000 and Re = 30000. Furthermore, the wakes of the baseline cases for Re = 61000 and Re = 83000 are more turbulent than the two lower Reynolds number cases. This is a reasonable result considering the increase in Reynolds numbers. We also observe that the wakes of the vortex generator installed cases are more turbulent than their baseline counterparts, at all Reynolds numbers.
The increased turbulent nature of the wake hints at an increase in heat transfer inside the cylinder wake: Increased turbulent mixing which is promoted by the presence of the vortex generators will combine high momentum fluid with low momentum fluid in regions near the endwall surfaces and the cylinders. As a result, heat transfer will be increased and cooling will be improved. The three dimensional streamlines given in the next subsection will support this statement.
7.1.2 Three Dimensional Streamlines

Figure 7.5 Re = 17000, Baseline and Vortex Generator installed cases, respectively
Figure 7.6 Re = 30000, Baseline and Vortex Generator installed cases, respectively
Figure 7.7 Re = 61000, Baseline and Vortex Generator installed cases, respectively
Figure 7.8 Re = 83000, Baseline and Vortex Generator installed cases, respectively

Figures 7.5 through 7.8 show that compared to the baseline case, the installation of the vortex generator pairs introduces a noticeable amount of turbulent mixing into the domain, for all Reynolds numbers. This is another indicator to the presence of increased momentum transport,
which leads to enhanced heat transfer levels in the domain. In addition, the wakes of the baseline cases for $\text{Re} = 61000$ and $\text{Re} = 83000$ are more turbulent when compared to the lower Reynolds number cases - a situation also seen in Chapter 7.1.1.

### 7.1.3 Heat Transfer Coefficient Contours on the Bottom and Top Surfaces

This section demonstrates the heat transfer distribution over the bottom and top surfaces of the cooling channel. As demonstrated by the following figures, it is noticed that the heat transfer coefficient levels show an increase compared to the baseline case when the vortex generators are installed, at each Reynolds number.
Figure 7.9 $Re = 17000$, Bottom Surface, Baseline and Vortex Generator installed cases, respectively.
Figure 7.10 Re = 30000, Bottom Surface, Baseline and Vortex Generator installed cases, respectively
Figure 7.11 Re = 61000, Bottom Surface, Baseline and Vortex Generator installed cases, respectively
Figure 7.12 $Re = 83000$, Bottom Surface, Baseline and Vortex Generator installed cases, respectively
As we expected from the increased turbulent nature of the wakes demonstrated in Chapter 7.1.1 and 7.1.2, cases at all Reynolds numbers demonstrate a significant amount of heat transfer increase over the majority of the lower surface when the vortex generators are installed. Certain portions of the lower wall immediately catch the eye:

1) The channels in between the cylinders of the first row and the wakes of these channels show a significant heat transfer increase due to the vortex generators.

2) The leading edges of the cylinders belonging to the second and third rows

3) The wakes of the cylinders belonging to the second and third rows

All of these locations are impingement zones for the streamlines which bring high momentum fluid towards the endwall. Due to the increased mixing of high and low momentum fluid compared to the baseline case, heat transfer is improved in these areas.
A similar situation exists for the top surface of each case, as demonstrated in Figures 7.13 through 7.16:

**Figure 7.13** $Re = 17000$, Top Surface, Baseline and Vortex Generator installed cases, respectively
Figure 7.14 Re = 30000, Top Surface, Baseline and Vortex Generator installed cases, respectively
Figure 7.15 Re = 61000, Top Surface, Baseline and Vortex Generator installed cases, respectively
Figure 7.16 Re = 83000, Top Surface, Baseline and Vortex Generator installed cases, respectively
7.1.4 Heat Transfer and Pressure Change Results

\[ N_{u_D} = \frac{hD}{k_{air}} \quad (7.1) \]

\[ f_{Factor} = \frac{p_{in} - p_{exit}}{\frac{1}{2}\rho u_{in}^2} \quad (7.2) \]

When the pairs of half height vortex generators are installed, the percentage improvements in heat transfer for each local (close to the vortex generators - a portion spanning 22 cm in the axial direction) region and large (a portion spanning 50 cm in the axial direction) region are given in Tables 7.1 through 7.4 below. Figures 7.17 through 7.19 show the local and large regions for reference (“cylinder” denotes the entire surface of each cylinder):

Figure 7.17 Top view of the bottom local and large areas
Figure 7.18 Isometric view of all the local areas

Figure 7.19 Isometric view of all the large areas
### Table 7.1 Heat Transfer Results, Re = 17000

<table>
<thead>
<tr>
<th></th>
<th>%Change in $\text{Nu}_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Bottom</td>
<td>11.04</td>
</tr>
<tr>
<td>Local Left</td>
<td>-2.29</td>
</tr>
<tr>
<td>Local Right</td>
<td>-1.11</td>
</tr>
<tr>
<td>Local Top</td>
<td>12.47</td>
</tr>
</tbody>
</table>

### Table 7.2 Heat Transfer Results, Re = 30000

<table>
<thead>
<tr>
<th></th>
<th>%Change in $\text{Nu}_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder 1</td>
<td>2.58</td>
</tr>
<tr>
<td>Cylinder 2</td>
<td>36.90</td>
</tr>
<tr>
<td>Cylinder 3</td>
<td>36.59</td>
</tr>
<tr>
<td>Cylinder 4</td>
<td>15.44</td>
</tr>
<tr>
<td>Large Bottom</td>
<td>21.87</td>
</tr>
<tr>
<td>Large Left</td>
<td>28.16</td>
</tr>
<tr>
<td>Large Right</td>
<td>27.71</td>
</tr>
<tr>
<td>Large Top</td>
<td>26.40</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>%Change in $\text{Nu}_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Bottom</td>
<td>4.25</td>
</tr>
<tr>
<td>Local Left</td>
<td>1.42</td>
</tr>
<tr>
<td>Local Right</td>
<td>2.41</td>
</tr>
<tr>
<td>Local Top</td>
<td>10.95</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>%Change in $\text{Nu}_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder 1</td>
<td>4.79</td>
</tr>
<tr>
<td>Cylinder 2</td>
<td>46.37</td>
</tr>
<tr>
<td>Cylinder 3</td>
<td>45.57</td>
</tr>
<tr>
<td>Cylinder 4</td>
<td>19.35</td>
</tr>
<tr>
<td>Large Bottom</td>
<td>15.71</td>
</tr>
<tr>
<td>Large Left</td>
<td>24.01</td>
</tr>
<tr>
<td>Large Right</td>
<td>29.88</td>
</tr>
<tr>
<td>Large Top</td>
<td>20.30</td>
</tr>
</tbody>
</table>
### Table 7.3 Heat Transfer Results, Re = 61000

<table>
<thead>
<tr>
<th></th>
<th>%Change in Nu&lt;sub&gt;D&lt;/sub&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Bottom</td>
<td>8.34</td>
</tr>
<tr>
<td>Local Left</td>
<td>0.95</td>
</tr>
<tr>
<td>Local Right</td>
<td>-3.76</td>
</tr>
<tr>
<td>Local Top</td>
<td>9.75</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>%Change in Nu&lt;sub&gt;D&lt;/sub&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder 1</td>
<td>-1.23</td>
</tr>
<tr>
<td>Cylinder 2</td>
<td>20.24</td>
</tr>
<tr>
<td>Cylinder 3</td>
<td>24.00</td>
</tr>
<tr>
<td>Cylinder 4</td>
<td>1.98</td>
</tr>
<tr>
<td>Large Bottom</td>
<td>18.30</td>
</tr>
<tr>
<td>Large Left</td>
<td>14.09</td>
</tr>
<tr>
<td>Large Right</td>
<td>9.69</td>
</tr>
<tr>
<td>Large Top</td>
<td>20.91</td>
</tr>
</tbody>
</table>

### Table 7.4 Heat Transfer Results, Re = 83000

<table>
<thead>
<tr>
<th></th>
<th>%Change in Nu&lt;sub&gt;D&lt;/sub&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Bottom</td>
<td>7.80</td>
</tr>
<tr>
<td>Local Left</td>
<td>-6.89</td>
</tr>
<tr>
<td>Local Right</td>
<td>-5.16</td>
</tr>
<tr>
<td>Local Top</td>
<td>7.56</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>%Change in Nu&lt;sub&gt;D&lt;/sub&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder 1</td>
<td>-6.02</td>
</tr>
<tr>
<td>Cylinder 2</td>
<td>13.11</td>
</tr>
<tr>
<td>Cylinder 3</td>
<td>10.15</td>
</tr>
<tr>
<td>Cylinder 4</td>
<td>-0.19</td>
</tr>
<tr>
<td>Large Bottom</td>
<td>15.90</td>
</tr>
<tr>
<td>Large Left</td>
<td>2.07</td>
</tr>
<tr>
<td>Large Right</td>
<td>4.55</td>
</tr>
<tr>
<td>Large Top</td>
<td>18.02</td>
</tr>
</tbody>
</table>
When we analyze the heat transfer results, we notice a couple of important points:

1) All Reynolds numbers generate increases in heat transfer for most of the selected wall surfaces.

2) Noticeable heat transfer increases are seen on the surfaces of the cylinders belonging to the second row, and the large surfaces (top, bottom, left and right) which cover the majority of the test section.

3) The significant effect of installing the VGs are more evident on the large portions, i.e we observe heat transfer increases up to 46%. Locally, we see improvements as well as decreases.

To summarize the results of the preceding tables, let's analyze Figure 7.20:

![Figure 7.20](image_url)

**Figure 7.20** Overall (Sum of all large surfaces and cylinders) heat transfer increase compared to the baseline case when VGs are installed
Figure 7.20 shows that our cooling channel configuration will generate promising improvements in heat transfer when half height vortex generators are installed, especially for $\text{Re} = 17000$ through $\text{Re} = 61000$: A heat transfer improvement varying between 13% and 27% is an important contribution to the gas turbine blade cooling channel flow.

All of the heat transfer improvements mentioned in this chapter are the result of the momentum transport increasing effect induced by the vortex generator. The streamlines which separate from the sharp edge of the vortex generator continue downstream in a vortical structure while promoting turbulent mixing in the entire domain. High momentum fluid located in the freestream is significantly mixed with lower momentum fluid in regions close to the endwalls and cylinders. As a result, heat transfer levels over these surfaces are enhanced.

![Graph showing % Increase in pressure loss vs Re number]

*Figure 7.21* Area Averaged Pressure Loss Coefficient variation
Figure 7.21 shows that when the area averaged inlet and exit pressures are used to calculate the pressure loss coefficient, we obtain increases up to 18.68%. This is consistent with the second law of thermodynamics: An increase in heat transfer is obtained, but this comes at the price of pressure loss across the channel. The heat transfer increase is more significant, since we are interested in the gas turbine channel cooling flow.

The preceding data suggests that it will be beneficial to use the half height vortex generators for cooling flows at Reynolds numbers ranging between 17000 and 83000. Therefore, Chapter 7.2 explains the details of the computational results of the Re = 30000 case in order to take a closer look at certain flow variables. In addition, the experimental results given in Chapter 7.4 are conducted with a Re = 61000 flow to provide an example of the effect of this setup in a laboratory environment.
7.2 Detailed Computational Fluid Dynamics Results for Re = 30000

7.2.1 Two Dimensional Streamlines

**Figure 7.22** Baseline Case - Two dimensional streamlines in the streamwise plane of symmetry

**Figure 7.23** Vortex Generator Installed Case - Two dimensional streamlines in the streamwise plane of symmetry
Figure 7.22 shows that the baseline case exhibits the expected horseshoe vortex roll-up behavior. Comparing Figures 7.22 and 7.23, we notice that the wake of the vortex generator installed case demonstrates a noticeable increase in turbulent structure. This is an indicator to improved momentum transport, which also translates as heat transfer enhancement.

### 7.2.2 Three Dimensional Streamlines

![Three Dimensional Streamlines](image)

**Figure 7.24** Baseline Case - Streamlines originating from rakes close to the lower surface
Figures 7.24 and 7.25 demonstrate that compared to the baseline case, the vortex generators introduce a significant amount of turbulent mixing into the domain. The streamlines which separate and break away from the sharp edge of the vortex generators mix the high momentum freestream fluid with the lower momentum fluid near the endwall and cylinder surfaces. As a result, the impingement of the high momentum containing turbulent flow structure on to the wall surfaces generate increased heat transfer.

Figures 7.26 and 7.27 demonstrate that the presence of the vortex generators promotes the turbulent mixing in the domain near the upper wall as well, more noticeably after the second row of cylinders.
Figure 7.26 Baseline Case - Streamlines originating from rakes close to the upper surface

Figure 7.27 Vortex Generator Installed Case - Streamlines originating from rakes close to the upper surface
As expected, the wake of the vortex generator installed case exhibits a significant increase in turbulent behavior. This flow structure supports the previously underlined statements.
about the physics that govern the vortex generator concept: Increased turbulence due to the presence of the vortex generators amplifies momentum transport towards the wall surfaces. Hence, heat transfer levels are enhanced.

7.2.3 Static Pressure Contours over the Bottom Surface

Figure 7.30 Baseline Case - Static Pressure contours over the bottom surface

Figure 7.31 Vortex Generator Installed Case - Static Pressure contours over the bottom surface
Figures 7.30 and 7.31 show that the vortex generators improve static pressure levels near the cylinder leading edges, as well as the vicinity of the vortex generators. The increased levels of static pressure are a result of the high momentum flow impingement in those areas. However, when the vortex generators are installed, as we noticed in Chapter 7.1.4, there is a decrease in the overall pressure loss between the inlet and exit of the domain. This is an expected result where the increase in heat transfer is accompanied by an increase in overall pressure loss.

Figures 7.32 and 7.33 show that vortex generators have a lower effect in changing pressure levels over the top surface, when compared to the bottom surface.
Figure 7.32 Baseline Case - Static Pressure contours over the top surface

Figure 7.33 Vortex Generator Installed Case - Static Pressure contours over the top surface
7.2.4 Heat Transfer Coefficient Contours over the Bottom and Top Surfaces

Figure 7.34 Baseline Case - Heat transfer coefficient contours over the bottom surface

Figure 7.35 Vortex Generator Installed Case - Heat transfer coefficient contours over the bottom surface
Figures 7.34 and 7.35 support the conclusions of the previous subsections: The installation of the vortex generator pairs noticeably improve the heat transfer levels over the bottom surface. The increased turbulent mixing provided by the vortex generators cause the flow to impinge on the endwall surface with high momentum, thereby increasing the heat transfer at those locations. Along with the results of Chapter 7.1.4, the heat transfer coefficient contours indicate that turbomachinery scale Reynolds numbers will generate significant increases in heat transfer.

Figures 7.36 and 7.37 demonstrate that the presence of the vortex generators introduce an improvement in heat transfer behavior over the top wall as well. The physical mechanism which leads to heat transfer improvements over the bottom wall work in the same way to improve the heat transfer distribution over the top wall. High momentum fluid which impinges on the surface due to the turbulent nature of the channel flow improves cooling at these impingement zones.
Figure 7.36 Baseline Case - Heat transfer coefficient contours over the top surface

Figure 7.37 Vortex Generator Installed Case - Heat transfer coefficient contours over the top surface
7.3 Comparison of the Baseline and Vortex Generator Installed Cases with the Plain Channel Case

In order to analyze the changes in flow behavior when installing the staggered rows of cylinders and the vortex generators into the plain duct, we included CFD results of the flow within the plain channel. This section contains the comparison of the plain channel with the baseline and vortex generator installed cases.

Table 7.5 compares the effect of adding the cylinders or adding the cylinders plus the vortex generator combination to the plain duct on the overall heat transfer:

<table>
<thead>
<tr>
<th></th>
<th>Re = 17000</th>
<th>Re = 30000</th>
<th>Re = 61000</th>
<th>Re = 83000</th>
</tr>
</thead>
<tbody>
<tr>
<td>% Increase when cylinders are added</td>
<td>126.5</td>
<td>110.5</td>
<td>113.8</td>
<td>115.9</td>
</tr>
<tr>
<td>% Increase when cylinders and VGs are added</td>
<td>182.0</td>
<td>167.1</td>
<td>142.0</td>
<td>130.6</td>
</tr>
</tbody>
</table>

Table 7.5 Effect of the cylinders and cylinders plus VG combination on the overall heat transfer

As expected, the addition of the cylinders generates a significant amount of heat transfer increase. The addition of the vortex generators improves the heat transfer even more. These improvements are due to the increased momentum transport induced by the cylinders. The vortex generators further add to the momentum transport to generate even higher heat transfer levels.
7.4 Experimental Results

The last stage of this thesis study involves the analysis of certain cases within the experimental setup described in Chapter 5. The experiments aim to verify the general trends of the findings of our CFD study. Various power settings and tunnel speeds have been studied, and the results of the 12 m/s, 21 V case are presented in detail as a representative of the experimental results.

7.4.1 Conduction Loss Analysis

![Conduction Loss vs. Voltage](image)

**Figure 7.38** Conduction loss variation with voltage change
Figure 7.39 Percentage of conduction loss compared to the total electrical power

Figure 7.38 depicts the value of conduction heat loss per unit area. The conduction loss thermocouples installed on the setup were used to obtain the temperatures for the conduction loss calculation. The conduction loss for each of these settings was measured over a period of 5 hours, and data was taken at 1 hour intervals in order to ensure the long term convergence of the conduction loss value. Figure 7.39 depicts the percentage value of the conduction loss compared to the total electrical power, which does not exceed 15% at any setting.
7.4.2 Adiabatic Wall Temperature Calculation

![Graph: 12 m/s q vs. ΔT]

**Figure 7.40** Variation of electrical power with adiabatic wall temperature correction

Figure 7.40 is obtained using the temperature difference between the average temperature of a small surface towards the right hand side of the heater, and the ambient temperature measured by the thermocouple probe. The plot is corrected by shifting it to its current position which passes through the origin, thus enabling the calculation of the adiabatic wall temperature.
7.4.3 Temperature Distribution Analysis

Figure 7.41 Schematic of the experimental setup and measurement zone (figure not to scale)

Figure 7.41 shows the location of the infrared window and the data extraction zone from which the temperature data of each pixel is taken. The data extraction zone is taken as such in order to avoid edge effects.
Figure 7.42 Plain Channel, Temperature Distribution, 21 V, 12 m/s

Figure 7.43 Baseline Case, Temperature Distribution, 21 V, 12 m/s
Figures 7.42 through 7.44 show the temperature distribution for each case over the data extraction zone. It is observed that the addition of the cylinders for the baseline case has altered the temperature distribution with convective effects. As expected from the trends shown by our CFD work, the addition of the vortex generators affected the distribution even further.
7.4.4 Overall Heat Transfer Coefficient Distribution Analysis

**Figure 7.45** Plain Channel, Heat Transfer Coefficient Distribution, 21 V, 12 m/s

h\_average = 69 W/m\(^2\).K

**Figure 7.46** Baseline Case, Heat Transfer Coefficient Distribution, 21 V, 12 m/s

h\_average = 94 W/m\(^2\).K

(h / h\(_0\))\_average = 1.36
The subscript "0" in the preceding figures refer to the value of the plain channel case. The location of the VGs is also given for easier visualization.

We observe that both the baseline case and vortex generator installed case show improvements in heat transfer levels compared to the plain channel case. The baseline case shows a 36% increase, while the vortex generator installed case shows a 50% increase. This 14% overall increase in heat transfer is accompanied by an 8% increase in pressure loss across the channel. These results confirm the general improvement trend observed in our CFD results: Similar to the effects seen in the CFD study, the addition of the vortex generators induce additional turbulent mixing into the domain. The increased turbulent mixing promotes the
movement of high momentum, cool freestream fluid towards the heater surface. Therefore, the impingement zones of this flow structure show increased heat transfer levels. Comparing Figures 7.46 and 7.47, it is observed that the wake of the vortex generator pair as well as the channel in between the middle row of cylinders are examples to this result. The physical process will be explained in detail in Chapter 7.5.

7.4.5 Local Heat Transfer Coefficient Analysis

This section analyzes the percentage change in the distribution of $h/h_0$ between the baseline and vortex generator installed cases along six lines which pass through crucial locations, as indicated in Figure 7.48:

![Figure 7.48](image)

**Figure 7.48** Six lines along which the heat transfer distribution is investigated
When analyze Figure 7.49, we observe that the addition of the vortex generators increase the $h/h_0$ values along the majority of the wake of the middle cylinder of the first row, as well as the passage between the two cylinders of the middle row. This is due to the effect of the impingement of high momentum containing fluid on the surface at these locations. In other words, the additional turbulent mixing induced by the vortex generators promotes additional heat transfer. This is especially noticeable by the rise in heat transfer levels up to 60% close to the vortex generator location.
Along line D, the majority of the flow domain shows increases up to 20%. The area close to the cylinder is affected by the impingement zone created by both pairs of vortex generators. As a result, the sharp rise in heat transfer is observed in the cylinder's wake. A similar observation is made for the increasing heat transfer region in the downstream portion of the channel. This location is another impingement zone for the high momentum containing streamlines.
Similar to the pattern observed along line E, the vicinity of the vortex generator wake exhibits significant heat transfer increases, varying between 15% and 30%. This is once again due to the increased momentum transport created by the flow separation and subsequent turbulent mixing induced by the vortex generator pairs. Line C is close to the mid-portion of the domain along the vertical axis, therefore the heat transfer levels are affected by both of the vortex generator pairs.
The distribution along line B underlines the presence of a large impingement zone upstream of the cylinder leading edge. This impingement zone is the result of high momentum flow movement induced by the vortex generators. Heat transfer levels show a sudden increase, rising as high as 50%. Downstream of the cylinder, a mean heat transfer increase of 5% is observed.
The heat transfer levels along line A show two zones of impingement and one of detachment. Near the vicinity of the lower cylinder of the second row, heat transfer is decreased by 15%, which indicates the presence of flow detaching from the surface and taking momentum away from that zone. Conversely, downstream and upstream of this location, significant heat transfer increases on the order of 20% are observed. These are locations which benefit from the attachment of high energy flow particles onto their surface.
Line F demonstrates the local distribution of heat transfer change along the vertical axis of the heater surface. When the vortex generators are installed, it is observed that the wake of the second row of cylinders show a mean increase in heat transfer ranging from 15% to 20%. This is due to the fact that the turbulent mixing promoted by the vortex generator pairs has continued downstream and brought high momentum fluid inside the wake of the second row of cylinders.

**Figure 7.54** Percentage change of $h/h_0$ on line F
7.5 Explanation of the Underlying Physics Related to the Vortex Generator Concept

This section focuses on explaining the underlying physics related to the vortex generator concept by making use of both our CFD and experimental results.

The first step in the explanation process is to look closely at how the streamlines behave in the presence of vortex generators, as indicated by CFD:

Figure 7.55 Streamlines seeded along the top edge of the vortex generator on the left

Figure 7.55 demonstrates the effect of installing the vortex generators has on the streamlines: The flow tends to separate and roll-up once it passes over the sharp top edge of the vortex generator. Then, the ensuing vortical structure continues downstream while promoting increased turbulent mixing between the freestream and boundary layer flow. This turbulent motion mixes the high momentum fluid located in the freestream with the lower momentum fluid near the endwalls and cylinder surfaces.
Figure 7.56 Streamlines seeded from a rake close to the junction of the vortex generator inner surface and lower wall surface

Figure 7.57 Closer look at Figure 7.56

Figures 7.56 and 7.57 demonstrate the streamline pattern in the vortical region induced by the vortex generator. The streamlines which separate and roll-up continue towards the lower
wall, then continue to move higher up inside the domain as they continue farther downstream. This movement pattern mixes the high momentum and low momentum fluids throughout the majority of the domain, due to its highly turbulent nature. Combined with the constricted passages generated by the blockage of the vortex generators, this flow structure generates large impingement zones on the endwalls and cylinder surfaces, thereby enabling increased heat transfer. Figures 7.58 and 7.59 approve this situation, and show zones of increased heat transfer when the vortex generators are present:

**Figure 7.58** Heat transfer contours along the bottom surface for the baseline and vortex generator installed cases, for CFD results obtained at Re = 61000 (12 m/s freestream velocity)
Figure 7.59 Heat transfer contours along the measurement surface for the baseline and vortex generator installed cases, for experimental results obtained at Re = 61000 (12 m/s freestream velocity)
To summarize, the mechanism of heat transfer enhancement is as follows:

1) The flow which separates and rolls up into a vortical structure as it passes over the sharp edge of the vortex generator, continues downstream in the form of longitudinal vortices.

2) These vortical structures generate large impingement zones over the lower wall, while promoting increased turbulent mixing with the freestream flow. The flow is also directed towards the surfaces of the cylinders located downstream, as well as the side and top walls. This flow pattern promotes the mixing of high and low momentum fluids throughout the majority of the channel.

3) The orientation and shape of the vortex generators also contribute to the movement pattern of the flow just described. The width to height ratio of the channel is also important, because too small a ratio (below 3) would negate some of the improvements generated by the vortex generator due to the dissipation of the concentrated turbulent mixing at larger channel heights.

4) Due to this flow structure, increased heat transfer is attained over the bottom wall as well as the cylinder surfaces and the top and side walls.

5) A loss in pressure across the channel inlet and exit accompanies the beneficial changes in heat transfer. Since the heat transfer improvements are larger than the loss in pressure, and the pressure loss levels are acceptable for internal cooling channel flow, the vortex generator concept is quite promising.
Chapter 8

Conclusion

This thesis study focused on investigating innovative methods to improve gas turbine cooling and performance. The study consists of a computational (CFD) part and an experimental part. Flow domains containing various suggestions for obtaining the desired results were created in GAMBIT. The flow simulations which used these computational regions were run in FLUENT. Once the CFD results reached an optimal level, experiments were conducted in the open loop wind tunnel of the Turbomachinery Aero-Heat Transfer Lab to verify the general trends of these computational findings in a real life atmosphere.

Two types of flow were of interest during this work: The gas turbine cooling channel flow and the flow around the NGV. A vertical circular cylinder was used to model a singular cooling channel pin-fin or the NGV blade. In a different set of runs, multiple circular cylinders were used to model a staggered array of circular pin-fins. The proposed modifications were the installation of an endwall fence upstream of the cylinder for the case of a singular pin-fin or the NGV blade. In the case of the staggered array of circular cylinders, common flow up oriented delta winglet type vortex generators were placed inside the gaps of the first row of cylinders.

To the best of our knowledge, our work is the first one to investigate the usage of vortex generators on internal cooling channel flow in gas turbine cooling. The results outlined in this chapter are applicable to aerospace science as well as other fields which use flows of similar Reynolds numbers and channel width to height ratios.
8.1 Endwall Fence Related Improvements

1) The upstream endwall fence successfully modifies the structure of the horseshoe vortex roll-up inside the boundary layer.

2) The time dependent characteristics of the flow were successfully captured.

3) In the case of the cooling channel, the endwall fence generates a mean heat transfer improvement of 10% downstream of the cylinder trailing edge over a region which is 7-8 cylinder diameters long. This is a significant result for internal cooling passages, where improved cooling of turbine blades allows their operational total temperature to be higher, which in turn yields better efficiency and thrust levels.

4) In the case of the flow around the NGV, the endwall fence shows local, spanwise distributed total pressure improvements. In addition, slight mass averaged total pressure based $C_p$ gains are observed inside the first 2 mm above the endwall.

5) All of the previously mentioned results were achieved with slight disturbances to the freestream flow. This is a good feature for the flow around the NGV where the mixture of boundary layer flow and freestream flow is not desired.
8.2 Vortex Generator Related Improvements

1) The common flow up oriented delta winglet type pairs of vortex generators successfully introduced a significant amount of turbulent mixing into the domain, due to the flow separation and roll-up induced by the sharp edges of the vortex generators.

2) The vortex generators direct the turbulent structured flow towards the surfaces of the cylinders and towards the bottom, top and side walls. This flow behavior also promotes the mixing of high momentum and low momentum fluid in regions near the endwalls and the cylinder surfaces.

3) As a consequence of the process described in Result 2, heat transfer improvements are observed along the bottom, top and side walls as well as the cylinder surfaces, in our CFD analysis.

4) All Reynolds numbers generate good heat transfer improvements. The vortex generator pairs are especially effective in generating beneficial heat transfer changes for the Reynolds numbers of 17000 through 61000.

5) For the Re = 17000 case, heat transfer increases up to 36% compared to the baseline case were seen on the surfaces of the cylinders belonging to the second (middle) row. Similarly, heat transfer improvements up to 46% were seen on the same cylinder surfaces for the Re = 30000 case.

6) The overall improvement in heat transfer is 25% for the Re = 17000 case, 27% for the Re = 30000 case, 13% for the Re = 61000 case and 7% for the Re = 83000 case.

7) All of the heat transfer improvements mentioned above are obtained at the price of increased area averaged pressure loss across the channel, which is in accord with the second law of thermodynamics: 12% for the Re = 17000 case, 13% for the Re = 30000 case, 19% for the Re = 61000 case and 18% for the Re = 83000 case.
8) A sample case using a tunnel speed of 12 m/s inside the open loop wind tunnel of the Turbomachinery Aero Heat Transfer Laboratory showed that the vortex generator installed case increases the heat transfer by 14% compared to the baseline case, over a surface covering the majority of the constant heat flux heater. An increase in pressure loss of 8% across the channel was observed alongside the heat transfer enhancements. Therefore, the general trend of positive change shown in the CFD analysis for the vortex generators was verified in an experimental environment. Local heat transfer distributions also show the positive contribution of the vortex generators along the entire length of the heater.
8.3 Recommendations for Future Studies

1) Both the upstream endwall fence and the common flow up oriented delta winglet type vortex generators deserve to be studied in more detail due to their effectiveness in improving heat transfer, especially the vortex generators.

2) The material strength of the upstream endwall fence and vortex generators must be investigated in detail before reaching the production step in manufacturing.

3) Erosion over time might be an issue for the vortex generators which have to be thin in order to be effective in generating heat transfer enhancements. Therefore, the material of which they are manufactured has to be carefully selected.

4) Different vortex generator geometries, positions and sizes must be investigated. An optimization based study could be very useful.

5) Time-dependent and/or LES based CFD computations of vortex generators could be useful.
References


Appendix A

Experimental Uncertainty Analysis

The parameter we are ultimately interested in is the heat transfer coefficient, \( h \). Therefore, our uncertainty analysis is based upon \( h \). The mathematical steps used to derive the final expression will be described in detail. In addition, the uncertainty value belonging to one of the experimented cases will be given as an example.

\[
\frac{i^2 R}{A_h} = \dot{q} = h(T_{wall} - T_\infty) \tag{A.1}
\]

Equation A.1 is modified to incorporate the conductive loss:

\[
\frac{i^2 R}{A_h} - k_{wall} \frac{T_{C1} - T_{C2}}{t_{wall}} = \dot{q} = h(T_{wall} - T_\infty) \tag{A.2}
\]

Solving for \( h \), we get:

\[
h = \frac{i^2 R}{A_h} \frac{T_{C1} - T_{C2}}{t_{wall}} \tag{A.3}
\]

The uncertainty of \( h \) depends on the uncertainty of the measured quantities included in A.3. Therefore, the differential value of \( h \) is calculated as follows:

\[
dh = \sqrt{\left( \frac{\partial h}{\partial R} \cdot dR \right)^2 + \left( \frac{\partial h}{\partial T} \cdot dT \right)^2 + \left( \frac{\partial h}{\partial T_{wall}} \cdot dT_{wall} \right)^2 + \left( \frac{\partial h}{\partial T_\infty} \cdot dT_\infty \right)^2 + \left( \frac{\partial h}{\partial T_{C1}} \cdot dT_{C1} \right)^2} \tag{A.4}
\]
\[ A = \left(\frac{\partial h}{\partial R} \cdot dR\right)^2 = \left(\frac{l^2dR}{A_h(T_{wall} - T_\infty)}\right)^2 \quad (A.5) \]

\[ B = \left(\frac{\partial h}{\partial l} \cdot dl\right)^2 = \left(\frac{2ldV}{A_h(T_{wall} - T_\infty)}\right)^2 \quad (A.6) \]

\[ C = \left(\frac{\partial h}{\partial T_{wall}} \cdot dT_{wall}\right)^2 = \left\{ \frac{I^2R}{A_h} - k_{wall} \frac{T_{C1} - T_{C2}}{t_{wall}} \right\} \quad \left(\frac{T_{wall} - T_\infty}{(T_{wall} - T_\infty)^2}\right)^2 \quad dT_{wall} \quad (A.7) \]

\[ D = \left(\frac{\partial h}{\partial T_\infty} \cdot dT_\infty\right)^2 = \left\{ \frac{I^2R}{A_h} - k_{wall} \frac{T_{C1} - T_{C2}}{t_{wall}} \right\} \quad \left(\frac{T_{wall} - T_\infty}{(T_{wall} - T_\infty)^2}\right)^2 \quad dT_\infty \quad (A.8) \]

\[ E = \left[ \frac{-k_{wall}}{l(T_{wall} - T_\infty)t_{wall}} dT_{C1} \right]^2 \quad (A.9) \]

\[ F = \left[ \frac{k_{wall}}{(T_{wall} - T_\infty)t_{wall}} dT_{C2} \right]^2 \quad (A.10) \]

\[ \text{Uncertainty} \% = 100\left(\frac{dh}{h}\right) = 100\left(\frac{\sqrt{A^2 + B^2 + C^2 + D^2 + E^2 + F^2}}{A_h - k_{wall} \frac{T_{C1} - T_{C2}}{t_{wall}}} \right) \quad (A.11) \]
Equation A.11 gives the percentage value of the experimental uncertainty. For example, the uncertainty for the baseline case run at 21 V and 12 m/s is 6.4%. The differential values which appear in terms A through F are the uncertainties of the devices used for measurement.
VITA
Zeki Özgür Gökçe

Zeki Özgür Gökçe was born on April 23rd, 1986 in Ankara, Turkey. He graduated with a Bachelor of Science in Aerospace Engineering degree from The Middle East Technical University (METU) on June 2008. He received his Master of Science in Aerospace Engineering degree from The Pennsylvania State University (PSU) on May 2010. He has been a graduate student at Penn State since August 2008.