NUMERICAL INVESTIGATION OF
BLADE LEADING EDGE CONTOURING BY
FILLET AND BASELINE CASE
OF A TURBINE VANE

- A comparative study of the effect on secondary flow

ANDREA MITRUS

Master of Science Thesis
Stockholm, Sweden 2012
NUMERICAL INVESTIGATION OF BLADE LEADING EDGE CONTOURING BY FILLET AND BASELINE CASE OF A TURBINE VANE

- A comparative study of the effect on secondary flow

Andrea Mitrus

Master of Science Thesis,
EGI-2012-045MSC EKV894

Department of Energy Technology
Division of Heat and Power Technology
Royal Institute of Technology
100 44 Stockholm, Sweden
ABSTRACT

The understanding of secondary flow behavior has become an important aspect in the design of modern gas turbines. Secondary flow gives rise to aerodynamic losses, distorts the thermal field and affects the flow conditions at the exit of a passage negatively. Therefore, reducing secondary flow is a major concern for efficiency improvement. Many passive control-methods have been suggested by turbine designers and researchers, and one very promising modification is blade leading edge contouring near the endwall. At the Division of Heat and Power Technology KTH, Stockholm, a detailed experimental investigation of three filleted nozzle guide vanes in an annular sector cascade has been performed, providing excellent experimental data for numerical validation of complex turbine flows.

Based on the above, a numerical study and aerodynamic investigation for a leading edge filleted vane and baseline vane has been performed. The potential effect of the leading edge fillet on flow structure and secondary losses has been evaluated based on a number of flow parameters, and computational predictions have been compared to experimental results.

The numerical investigation has shown some differences in the flow behavior between the filleted and baseline case. All results indicate that the fillet affects the flow structure in regions close to the hub endwall. It shifts the position of vortices and loss core. However, the overall effect on reducing secondary losses downstream of the passage is insignificant. Additionally, the numerical results show good qualitative agreement with experimental results.
# TABLE OF CONTENTS

ABSTRACT ......................................................................................................................... I

TABLE OF CONTENTS ....................................................................................................... III

LIST OF FIGURES ............................................................................................................. V

LIST OF TABLES ............................................................................................................... VII

NOMENCLATURE ........................................................................................................ VIII

1 INTRODUCTION........................................................................................................ 1
  1.1 Objective ............................................................................................................... 2
  1.2 Methodology ........................................................................................................ 2

2 BACKGROUND.............................................................................................................. 3
  2.1 Secondary flow structure .................................................................................. 3
    2.1.1 Endwall crossflow ................................................................................... 4
    2.1.2 Horseshoe vortex .................................................................................... 5
    2.1.3 Passage vortex and induced vortices ..................................................... 5
  2.2 Aerodynamic losses ............................................................................................ 7
  2.3 Methods of reducing secondary losses ............................................................. 8
  2.4 Testing facilities for investigation of turbomachinery flow ............................... 10
    2.4.1 Annular sector cascade facility at KTH ............................................... 11
  2.5 Outcomes of former investigations ................................................................ 12
  2.6 Computational fluid dynamics ........................................................................ 13
    2.6.1 Mathematics of CFD ............................................................................. 14
    2.6.2 Limitations of CFD ................................................................................ 14

3 CFD MODELING .......................................................................................................... 16
  3.1 Geometry ............................................................................................................. 16
  3.2 Mesh .................................................................................................................... 18
  3.3 Physics of the model .......................................................................................... 22

4 AERODYNAMIC INVESTIGATION OF FLOW FIELD QUALITY ............................. 24
  4.1 Load distribution ................................................................................................. 26
  4.2 Flow angles ......................................................................................................... 31
  4.3 Vorticity ............................................................................................................... 35
    4.3.1 Formation of horseshoe vortex ............................................................. 35
    4.3.2 Vortex distribution in downstream plane ............................................. 37
  4.4 Aerodynamic loss ............................................................................................... 40

5 CONCLUSIONS AND FUTURE WORK ................................................................... 43
LIST OF FIGURES

Figure 1.1: (a) Schematic of gas turbine (b) Gas turbine cycle efficiency (Cohen 1987) ................................................................. 1
Figure 2.1: Boundary layer velocity profile (Bartl 2010) ................................................................. 3
Figure 2.2: (a) MS flow streamlines around vanes (b) Pitchwise pressure gradient from high pressure (red) to low pressure (green) ......................................................... 4
Figure 2.3: Endwall crossflow ........................................................................................................... 4
Figure 2.4: Horse shoe vortex formation just before leading edge (Bartl 2010) ......................... 5
Figure 2.5: Secondary flow model according to Langston (Langston 1980) .............................. 6
Figure 2.6: Secondary flow model according to Wang (Wang 1997) ............................................. 6
Figure 2.7: Secondary flow model according to Goldstein and Spores (Goldstein 1988) ................. 7
Figure 2.8: Leading edge contouring with (a) fillet (b) large bulb (c) small bulb (Becz 2004) ................................................................................................................................. 9
Figure 2.9: (a) Investigated NGV with tip endwall contouring and fillet (b) Non-axisymmetric endwall contouring (Acharya 2006) ........................................................................ 9
Figure 2.10: Photo of ASC facility at KTH (Saha 2012) ................................................................. 11
Figure 2.11: Axial section view of LE fillet and front part of NGV (Saha 2012) ...................... 11
Figure 3.1: Radial section view of CAD model .............................................................................. 16
Figure 3.2: (a) Cascade model before change (b) Cascade model after change (c) Ideal model ........................................................................................................................................ 17
Figure 3.3: Block topology of cascade model .............................................................................. 18
Figure 3.4: Turbulent boundary layer (adapted from ANSYS 1) ....................................................... 20
Figure 3.5: Surface mesh of cascade model ...................................................................................... 21
Figure 3.6: Surface mesh of ideal models ......................................................................................... 21
Figure 3.7: Inlet total pressure profile 55.7% cax,hub upstream of the LE ........................................ 22
Figure 4.1: Location of data extraction points for comparison with experiment .............. 25
Figure 4.2: Inlet plane of cascade: y- and z- coordinates vs. r- and Φ- coordinates 25
Figure 4.3: (a) Blade loading at 25% span (b) Pressure contour on NGV0 ............................... 27
Figure 4.4: (a) Extensions of fillet: (b) 10% span (c) 5% span ................................................... 28
Figure 4.5: Blade loading: (a) 10% span (b) 5% span ................................................................. 30
Figure 4.6: Pitch-averaged exit flow angle ................................................................................... 31
Figure 4.7: Pitch-averaged exit flow angle in secondary flow region ................................... 32
Figure 4.8: Pitch-averaged exit flow angle in secondary flow region compared to experiment .................................................................................................................... 32
Figure 4.9: Pitch-averaged exit flow angle comparison with experiment .......................... 33
Figure 4.10: Pitch-averaged radial flow angle ............................................................................... 34
Figure 4.11: Meridional view: velocity vectors with gridlines .................................................. 34
Figure 4.12: Velocity streamlines baseline case: (a) Hub and blade (b) blade-to-blade cut (hub) ...................................................................................................................... 35
Figure 4.13: Vorticity contour baseline case: Legs of HS vortex seen from leading edge plane ................................................................................................................................. 36
Figure 4.14: Velocity streamlines filleted case: (a) Hub and blade (b) blade-to-blade cut (hub) ................................................................................................................................. 36

V
Figure 4.15: Vorticity contour filleted case: No HS vortex formation in leading edge plane .................................................................................................................................................. 37
Figure 4.16: Vorticity contour filleted case: Legs of HS vortex on the fillet (0% \( c_{ax,hub} \)) ........................................................................................................................................................................... 37
Figure 4.17: Vorticity contour plot downstream plane 107.1% of \( c_{ax,hub} \): Baseline case ........................................................................................................................................................................................................ 38
Figure 4.18: Vorticity contour plot downstream plane 107.1% of \( c_{ax,hub} \): Filleted case ........................................................................................................................................................................................................ 38
Figure 4.19: Axial vorticity at 10% span ........................................................................................................................................................................................................... 39
Figure 4.20: Axial vorticity at 5% span ........................................................................................................................................................................................................... 39
Figure 4.21: Vorticity distribution in downstream plane for CFD and experiment .... 40
Figure 4.22: Normalized total pressure contour in downstream plane (a) Baseline (b) Fillet ........................................................................................................................................................................................................ 41
Figure 4.23: Mass-averaged kinetic energy loss close to hub endwall .......... 42
Figure 4.24: Mass-averaged kinetic energy loss close to hub endwall for CFD and experimental results ........................................................................................................................................................................................................ 42
Figure 7.1: Settings for smoothing pre-mesh ........................................................... 47
Figure 7.2: Pre-mesh: determinant values before smooth ........................................ 47
Figure 7.3: Pre-mesh: determinant values after smooth ........................................... 47
Figure 7.4: Pre-mesh: volume values before smooth ............................................... 48
Figure 7.5: Pre-mesh: volume values after smooth .................................................. 48
Figure 7.6: Pre-mesh: minimum angle values before smooth .................................. 48
Figure 7.7: Pre-mesh: minimum angle values after smooth .................................... 48
Figure 7.8: Settings for smoothing mesh .................................................................. 49
Figure 7.9: Mesh: determinant values before smooth ............................................. 49
Figure 7.10: Mesh: determinant values after smooth .............................................. 49
Figure 7.11: Mesh: volume values before smooth .................................................. 50
Figure 7.12: Mesh: volume values after smooth ..................................................... 50
Figure 7.13: Mesh: minimum angle values before smooth ..................................... 50
Figure 7.14: Mesh: minimum angle values after smooth ....................................... 50
Figure 8.1: Baseline case: Loading ......................................................................... 51
Figure 8.2: Baseline case: \( M_{iso2} \) at hub endwall .................................................. 51
Figure 8.3: Filleted case: Loading ......................................................................... 52
Figure 8.4: Filleted case: \( M_{iso2} \) at hub endwall .................................................. 53
Figure 8.5: Cascade case: Loading ........................................................................ 54
Figure 8.6: Filleted case: \( M_{iso2} \) at hub endwall .................................................. 54
Figure 9.1: Load distribution 50% span ................................................................. 55
Figure 9.2: Load distribution 75% span ................................................................. 55
Figure 9.3: CFD: Load 75% span ........................................................................... 56
Figure 9.4: CFD: Load 50% span ........................................................................... 56
Figure 9.5: CFD: Load 25% span ........................................................................... 57
Figure 9.6: CFD: Load 13% span ........................................................................... 57
Figure 10.1: Velocity components for calculating exit flow angle and radial flow angle (adapted from ANSYS 2) ........................................................................................................................................................................................................ 58
Figure 11.1: (a) Axial vorticity scale (b) Downstream viewpoint of extraction-plane within passage ........................................................................................................................................................................................................ 59
Figure 11.2: Axial vorticity at 0% $c_{ax,hub}$ ................................................................. 59
Figure 11.3: Axial vorticity at 20% $c_{ax,hub}$ ................................................................. 60
Figure 11.4: Axial vorticity at 60% $c_{ax,hub}$ ................................................................. 60
Figure 11.5: Axial vorticity at 80% $c_{ax,hub}$ ................................................................. 61
Figure 11.6: Axial vorticity at 90% $c_{ax,hub}$ ................................................................. 61
Figure 11.7: Axial vorticity at 95% $c_{ax,hub}$ ................................................................. 62
Figure 11.8: The resulting vortices in the downstream plane at 107.1% $c_{ax,hub}$ ........ 62
Figure 11.9: Baseline case: whole downstream plane with spanlines ...................... 63
Figure 11.10: Filleted case: whole downstream plane with spanlines ...................... 63
Figure 11.11: Axial vorticity at 25% span ................................................................. 64
Figure 11.12: Axial vorticity at 50% span ................................................................. 64
Figure 11.13: Axial vorticity at 75% span ................................................................. 65
Figure 11.14: Axial vorticity at 90% span ................................................................. 65
Figure 12.1: Mass-averaged kinetic energy loss distribution CFD ......................... 66
Figure 12.2: Mass-averaged kinetic energy loss distribution experiment .............. 67
Figure 12.3: Mass-averaged kinetic energy loss distribution: Comparison CFD and experiment ........................................................................................................ 67

LIST OF TABLES

Table 3.1: General settings used for CFD models .................................................... 23
Table 4.1: $M_{iso3}$ at hub endwall .............................................................................. 24
Table 8.1: Baseline case: Mesh density and averaged $y^+$ value ......................... 52
Table 8.2: Filleted case: Mesh density and averaged $y^+$ value ......................... 53
Table 8.3: Cascade case: Mesh density and averaged $y^+$ value ...................... 54
## NOMENCLATURE

### Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Chord length</td>
<td>[mm]</td>
</tr>
<tr>
<td>h</td>
<td>Height</td>
<td>[mm]</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
<td>[-]</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
<td>[Pa]</td>
</tr>
<tr>
<td>r</td>
<td>Radius</td>
<td>[mm]</td>
</tr>
<tr>
<td>s</td>
<td>Pitch</td>
<td>[mm]</td>
</tr>
<tr>
<td>u</td>
<td>Velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>x</td>
<td>x-coordinate</td>
<td>[mm]</td>
</tr>
<tr>
<td>y</td>
<td>y-coordinate</td>
<td>[mm]</td>
</tr>
<tr>
<td>z</td>
<td>z-coordinate</td>
<td>[mm]</td>
</tr>
<tr>
<td>α</td>
<td>Exit flow angle</td>
<td>[°]</td>
</tr>
<tr>
<td>β</td>
<td>Radial flow angle</td>
<td>[°]</td>
</tr>
<tr>
<td>ζ</td>
<td>Energy loss coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>Φ</td>
<td>Coordinate in CCS</td>
<td>[°]</td>
</tr>
<tr>
<td>κ</td>
<td>Specific heat ratio</td>
<td>[-]</td>
</tr>
<tr>
<td>θ</td>
<td>Kinematic viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>ρ</td>
<td>Density</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>τ</td>
<td>Shear stress</td>
<td>[Pa]</td>
</tr>
<tr>
<td>Ω</td>
<td>Circumferential direction</td>
<td>[-]</td>
</tr>
<tr>
<td>ω</td>
<td>Vorticity</td>
<td>[1/s]</td>
</tr>
</tbody>
</table>

### Subscripts

- **1**: Upstream plane position
- **2**: Downstream plane position
- **3**: Operating point position
- **ax**: Axial reference
- **BL**: Boundary layer
- **b2b**: Blade to blade
- **hub**: Hub
- **iso**: Isentropic
Abbreviations

kin  Kinetic
MS   Mainstream
PS   Pressure side
SS   Suction side
s    Static
sp   Spanwise
w    Wall
x    x-direction
y    y-direction
z    z-direction

ASC  Annular Sector Cascade
BL   Boundary Layer
CAD  Computer Aided Design
CCS  Cylindrical Coordinate System
CFD  Computational Fluid Dynamics
CV   Corner Vortex
HS   Horseshoe
LE   Leading Edge
MS   Mainstream
MW   Mega Watt
NGV  Nozzle Guide Vane
PV   Passage Vortex
PS   Pressure Side
PLeCV Pressure Side Leading Edge Corner Vortex
SLeCV Suction Side Leading Edge Corner Vortex
SS   Suction Side
SST  Shear Stress Transport
TE   Trailing Edge
The role of thermal turbomachines is significant in the production of mechanical and electrical energy. Gas turbines account for approximately 17% of the world’s power generating capacity and give propulsion to most modern aircrafts (Strand 2007).

The schematic of a simple gas turbine is shown in Figure 1.1 (a). The working principle is based on the Brayton cycle where air at ambient conditions is compressed and delivered into the combustion chamber where fuel is burned, resulting in a hot pressurized gas. The gas is expanded in the turbine to generate work.

![Schematic of gas turbine](image)

**Figure 1.1:** (a) Schematic of gas turbine (b) Gas turbine cycle efficiency (Cohen 1987)

The turbine efficiency stands in direct proportion to the turbine inlet temperature. As seen in Figure 1.1 (b), higher temperatures give better cycle efficiency. The problem is that the stator, or nozzle guide vane (NGV), now experiences temperatures above its melting point. Film cooling is a common method to reduce the thermal load on airfoil and endwall surfaces. The cooling air is bled from the compressor and injected through small holes to the exposed surfaces providing a protective film. However, secondary flows can work contra productively by lifting the film-cooling jets away from the surfaces and thereby reduce their effectiveness. Additionally, they can cause surface heat transfer to be increased even further by transporting high temperature gas from the main flow to airfoils and endwalls. The third problem associated to secondary flow is that it gets its energy from the main flow, resulting in high aerodynamic losses.

Secondary flow can account for as much as for 35-40% of the total aerodynamic loss and thermal loading in a turbine passage and it is thereby of great interest to reduce this phenomenon (Acharya 2006). Extensive research has been conducted in the field, and blade leading edge contouring by fillet is one of the methods that have shown great potential on reducing secondary flow.
Introduction

1.1 Objective

The understanding of secondary flow has become important in the design of modern gas turbines since many undesirable effects are associated to it. Computational Fluid Dynamics (CFD) is a computer-based tool that can serve as an important complement to experimentally conducted studies and give valuable insights of complex flow phenomena. An advantage of CFD codes is that extremely large volumes of results can be produced and extracted from locations where controlled experiments are impossible to perform. Recent experimental testing has been performed in an annular sector cascade (ASC) cold flow test rig, examining the influence of a fillet on secondary flow development. The present study aims on contributing to this research by a numerical investigation based on the same setup used in the experiment.

With this background, three particular objectives can be formulated for this study:

- Evaluate the influence of the leading edge (LE) fillet on secondary flow
- Produce and extract a wide range of results which were not possible to obtain via experiments
- Use experimental data to validate the CFD models

1.2 Methodology

The study is divided into three phases. In the first phase, background knowledge is gathered through a literary review and the commercial software that is used, ANSYS\(^1\) ICEM and CFX, is studied.

The second phase is dedicated to numerical simulations of both cascade and periodic setup. The procedure follows the steps below:

1) Import and adjustment of CAD geometry
2) Mesh generation
3) Simulations of fluid flow
4) Processing of results

The CAD model of the ASC test rig is imported to ANSYS ICEM where the geometry is adjusted and meshes are generated. Simulation and visualization of results is accomplished in ANSYS CFX and with the help of MATLAB\(^2\) the results are processed to evaluate the flow quality.

In the last phase the results are analyzed, compared to experimental findings and conclusions are drawn.

\(^1\) Version 12.1
\(^2\) Version 7.12.0.635 (R2011a)
2 BACKGROUND

This chapter is intended to give a background to the main topics of the present study and important notions necessary for understanding the discussion of results are introduced.

2.1 Secondary flow structure

The understanding of secondary flow behavior has become increasingly important in the design of modern gas turbines. Secondary flow gives rise to aerodynamical losses, distorts the thermal field and affects the flow conditions at the exit of a passage negatively. Losses associated to the secondary flow will in this study be referred to as secondary losses.

Different models describing secondary flow have been presented throughout the years, with varying reports on the vortex formations occurring inside the passage. However, the basic secondary flow phenomena such as endwall crossflow, horseshoe (HS) vortex and passage vortex (PV) are generally agreed upon and will be presented in the following sections. The differences between the models are therefore mainly regarding the interaction schemes of these basic vortices and the supplementary vortices they might induce. Three models of different complexity will be discussed briefly.

Secondary flow phenomena occurs in the passage between two vanes/blades where the flow can be divided into the mainstream (MS) flow far away from passage endwalls and flow close to passage endwalls. The MS flow essentially follows the ideal flow behavior and may be considered as two-dimensional with streamwise and tangential pressure gradients. However, the flow near the endwall boundary layer (BL) is strongly three-dimensional and has, in addition to the streamwise and tangential pressure gradients, also a spanwise velocity/pressure gradient (Figure 2.1).

![Figure 2.1: Boundary layer velocity profile (Bartl 2010)](image)

3 Two-dimensional assumption of MS flow only valid for flow in linear cascades where no radial pressure gradient is present (see Chapter 2.4)
2.1.1 Endwall crossflow

As can be seen in Figure 2.2, the MS flow passing through a passage turns with the curvature of the vane and sets up a pitchwise pressure gradient from the pressure side (PS) to the suction side (SS).

\[ \frac{p_{PS} - p_{SS}}{\text{pitch}} \approx \frac{dp}{dr} = \rho \frac{u_{MS}^2}{r_{MS}} = \rho \frac{u_{BL}^2}{r_{BL}} \]  

(2.1)

Figure 2.2: (a) MS flow streamlines around vanes (b) Pitchwise pressure gradient from high pressure (red) to low pressure (green)

Equation 2.1 shows the relationship between this pressure gradient \( dp \), the velocity \( u \), the density \( \rho \) and the streamline curvature (represented by radius \( r \)). According to the equation, since the velocity in the BL is smaller than the MS velocity, also the radius of curvature for the BL flow is smaller than for the MS flow (Figure 2.3). This leads to secondary flow from PS to SS, in crosswise direction to the main flow, also known as endwall crossflow.

Figure 2.3: Endwall crossflow
2.1.2 Horseshoe vortex

The primary source of the complex vortex flows developing in a turbine passage is the HS vortex. When the incoming BL approaches the vane/blade LE stagnation line, the total pressure gradient becomes a static pressure gradient with lower pressure close to the endwall. This pressure gradient causes a vortex roll-up just before the LE (Figure 2.4). The downwards directed flow reaches the endwall, flows upstream and separates from the surface at the saddle point. The vortex center is located between the saddle point and the LE. The exact locations of the saddle point and the HS vortex center are determined by the incoming BL thickness and the curvature of the LE (Acharya 2006).

To elude the obstacle presented by the vane, the HS vortex splits into a PS leg and a SS leg, with opposite senses of rotation. The SS leg will propagate downstream through the passage staying close to the SS of the vane. The PS leg, however, is drawn from the PS to the SS of the passage by the pitchwise pressure gradient and the endwall crossflow.

![Figure 2.4: Horse shoe vortex formation just before leading edge (Bartl 2010)](image)

2.1.3 Passage vortex and induced vortices

As the PS leg of the HS vortex travels towards the SS, it combines with the endwall crossflow, growing in size and intensity to generate the PV.

A basic model of the secondary flow was presented by Langston (1980), showing the main vortices that have been discussed so far (Figure 2.5). According to Langston, the SS leg of the HS vortex (also referred to as counter vortex) rotates around the larger PV as a planet around the sun. As a consequence, the position of the SS leg relative to the PV might be different than that shown in the figure.
Background

Figure 2.5: Secondary flow model according to Langston (Langston 1980)

Since the Langston model, many studies have been performed giving a more complex picture of the phenomena occurring inside a turbine passage. Two examples are the models by Wang et al. (1997), Figure 2.6, and Goldstein and Spores (1988), Figure 2.7. Their main difference is the position of the SS-leg of the HS vortex in relation to the PV.

According to Wang, the PV gradually lifts above the endwall and the SS leg of the HS vortex wraps around it as they travel through the passage along the SS. At the passage exit, the PV will have moved towards the midspan and the SS leg of the HS vortex is located beneath it (Wang 1997). The Goldstein model, on the other hand, suggests that the SS-leg of the HS vortex is located above the PV (Goldstein 1988). This is also in agreement with the results obtained from the CFD simulations performed in this study.

Figure 2.6: Secondary flow model according to Wang (Wang 1997)
Apart from the main vortices already mentioned, there are also a few induced vortices associated to secondary flow that are present in both models. The HS vortex induces two small vortices identified as the *pressure-side leading edge corner vortex* (PLeCV) and *suction-side leading edge corner vortex* (SLeCV). Basically these small vortices follow the same pattern through the passage as the leg vortices they are associated with, but with opposite sense of rotation (Acharya 2006).

About half way downstream in the passage, at the junction between endwall and blade, two vortices are formed denoted *pressure-side corner vortex* and *suction-side corner vortex*. They both have the same sense of rotation as the SS leg of the HS vortex (Acharya 2006) and for simplicity reasons they will just be referred to as *Corner vortices* (CV).

Wang et al. (1997) also identified another vortex, the so-called *Wall vortex*. It is induced by the strong PV, at the same location as the PV is formed, and travels above it towards the passage exit.

### 2.2 Aerodynamic losses

Losses are commonly categorized as “profile loss”, “secondary loss” and “leakage loss”. However, this distinction is not always straightforward since these loss mechanisms are rarely independent of each other. The relative magnitudes of these losses vary depending on machine, but generally speaking each can be taken to account for one third of the total loss (Denton 1993). The vortex structures associated to the *secondary loss* were described in chapter 2.1. The aerodynamic losses arise from dissipation of the kinetic energy from rotation of the vortices, little of which is recovered in the following blade rows.

Tip *leakage loss* is caused by the leakage of flow over the tips of rotor blades and the hub clearance of stator blades. This is not an issue in the present study and will therefore not be discussed here.
Profile losses are directly related to the flow around the blade profile and its boundary layer. Trailing edge (TE) losses and shock losses are often included in the profile loss. Boundary layer separation presents a major source of profile losses in a blade passage (Acharya 2006). It may occur in regions on the blade of approximately constant static pressure or increasing static pressure. Since the normal behavior of flow is to stream towards lower pressure, this may result in back flow close to the wall and the formation of a separation bubble, in which the kinetic energy is dissipated and losses are raised. As a consequence, the risk of separation is smaller for an accelerated flow.

The wake region behind a blade is a direct manifestation of the TE losses, which are highly dependent on the boundary layers on PS and SS just upstream of the TE, and the TE thickness. There are two important phenomena leading to TE losses, one is the base pressure and the other one is the formation of the von Karman vortex street. The base pressure is the pressure acting on the TE and it is generally lower than the pressure in the free stream. Because of this pressure gradient, the flow streams crosswise. The von Karman vortex street forms due to the boundary layer separation close to the TE, causing a system of vortices and therefore a loss of total pressure. In this study, the vortices formed just downstream of the TE are referred to as TE shed vortex.

When the flow approaches supersonic values, usually close to the TE, shock waves occur. These are characterized by abrupt deceleration and change of density, static temperature and pressure (Putz 2010).

2.3 Methods of reducing secondary losses

It seems that a reduction in strength and size of the PV or HS vortex could be very beneficial for reducing the aerodynamic losses and endwall thermal loading. Numerous experimental and numerical investigations have been performed in attempts to control secondary flow through different structural modifications.

Leading edge fillet, or leading edge contouring near the endwall, is the main topic of this study, but endwall profiling and coolant film injection through shaped holes in the endwall will also be discussed briefly.

Fillets come in many shapes and sizes and can generally be divided into two categories (fillets and bulbs) depending on how they blend with the endwall and blade surfaces (Acharya 2006). The thickness of the fillet profile reduces to zero as it extends from the blade surface to the endwall. An example of a fillet can be seen in Figure 2.8 (a). The bulb profile blends with the blade surface as it wraps around the LE, but meets the endwall with a finite thickness. This is illustrated in Figure 2.8 (b) and (c). Leading edge fillets are placed in the stagnation region at the junction of the LE and the endwall with the scope to reduce or eliminate the formation of the HS vortex. In consequence this should reduce the downstream development of the PV.
Background

The bulb profiles, on the other hand, aim on intensifying the SS-leg of the HS vortex and thereby reduce the strength of the PV.

![Figure 2.8: Leading edge contouring with (a) fillet (b) large bulb (c) small bulb (Becz 2004)](image)

There are two types of endwall profiling, or endwall contouring; axisymmetric and non-axisymmetric. The first only has axial variations in the profile whereas the latter has both axial and pitchwise.

The function of the axisymmetric profile is to accelerate the BL fluid through the passage in order to reduce the BL thickness and thereby suppress the growth of secondary flows on the endwalls. The profiling is employed on either tip or hub endwall, with the height of the profile increasing over a smooth curve from the LE to the TE (Acharya 2006). In this way the passage cross-section is narrowed and the flow is accelerated. The investigated Nozzle Guide Vane (NGV) of this study has a tip platform with an axisymmetric profile as can be seen in Figure 2.9 (a).

![Figure 2.9: (a) Investigated NGV with tip endwall contouring and fillet (b) Non-axisymmetric endwall contouring (Acharya 2006)](image)
Background

The purpose of a non-axisymmetric profile is to decrease the endwall region static pressure on the PS and increase it near the SS (Acharya 2006). This will reduce the pitchwise pressure gradient and the strength of the endwall crossflow at the endwalls. This is achieved by increasing the endwall height near the passage PS and decreasing it near the SS (Figure 2.9 b).

As mentioned in the introduction, coolant film injection is a common way of protecting endwall and blade surfaces from the hot gases. Unfortunately, secondary flow, especially the PS-leg vortex and SS-leg vortex, usually lift the coolant film away, causing enhanced thermal loading on these parts. However, since the coolant jets have a direct impact on the endwall pressure field and cross-pitch flow, there is a potential to use endwall film injection to manipulate the BL in a desirable way and hence reduce secondary flows (Acharya 2006).

2.4 Testing facilities for investigation of turbomachinery flow

Performing experimental research on full-scale machines under normal operating conditions is not only extremely costly, but also presents a great challenge on installation of the measurement equipment that needs to withstand very high temperatures. Hence, using cold test facilities of different models is a good solution. In order to recreate the operating conditions of a real gas turbine, similarity principles (geometry, Mach number, Reynolds number etc.) are applied. Generally there are three different models used: linear cascade, annular cascade and annular sector cascade.

Linear cascades are the most commonly used models for conducting experimental testing due to their economic advantages, geometric simplicity and easy setup. The finite numbers of vanes are linearly aligned separated by constant pitch. The main disadvantages related to linear cascades are the non-established radial pressure gradients, resulting in two-dimensional flow conditions, and poor flow periodicity due to the introduction of sidewalls.

Annular cascades are the best representation of real machines, with a three-dimensional flow field and the possibility to test exact replicas of three-dimensional vane/blade profiles. The drawbacks are mainly the high capital and operating costs and the reduced accessibility for instrumentation.

The annular sector cascade (ASC), combines the advantages of both previous models. A radial pressure gradient is established, actual engine component profiles can be tested and size and running costs are reduced compared to the full annular cascade. On the negative part, the aspect of poor flow periodicity is introduced due to the presence of sidewalls.
2.4.1 Annular sector cascade facility at KTH

The annular sector cascade facility was installed in 1998 at the Division of Heat and Power Technology at KTH, Stockholm. Over the years, extensive experimental investigations have been performed with the common scope to improve the efficiency of modern gas turbines. The test section is a 36° annular sector with three NGVs and two sidewalls, seen in Figure 2.10. Air is supplied by a twin screw compressor powered by a 1 MW electric motor, and an air cooler is used to cool the air down to 30°C. The maximum continuous airflow is approximately 4.7 kg/s at 4 bars (Saha 2012).

Figure 2.10: Photo of ASC facility at KTH (Saha 2012)

Recent experimental testing has been performed in the ASC, examining the influence of a fillet on secondary flow development. The present study aims on contributing to this research by a numerical investigation based on the same setup used in the experiment.

The height of the fillet varies linearly from the vane surface to the hub endwall, and a schematic picture of it can be seen in Figure 2.11. The height of the fillet is around 10% of the full LE height and the upstream extension is about 30% of the axial chord (Saha 2012). The fillet has an asymmetric profile where the SS extension is more pronounced than the PS extension.

Figure 2.11: Axial section view of LE fillet and front part of NGV (Saha 2012)
2.5 Outcomes of former investigations

Many investigations, both experimental and numerical, have been performed on leading edge contouring and its effect on the development of secondary flows in vane passages. The focus has foremost been to reduce the aerodynamic losses and secondly to reduce the thermal loading.

As already mentioned most investigations have been performed in linear cascades where there is no radial pressure gradient established, which is an important factor for the development of secondary flow. Based on this, Saha (2012) performed an experimental investigation in the ASC described earlier with a geometric replica of a modern gas turbine NGV with and without a LE fillet. Since secondary flows also depend strongly on the incoming BL, the tests were performed with different prehistories. Two different turbulence grids were used to obtain different upstream conditions. With one turbulence grid, an inlet BL of approximately 8-10% of the inlet span was obtained. This was taken as a design criterion for the fillet height. With the other turbulence grid a BL thickness of 14-16% was accomplished. The general conclusion from the investigation was that the LE fillet had no effect on reducing secondary loss. On the other hand, different inlet conditions had a direct impact on the losses. More detailed results from the investigation will be presented in chapter 4.

The following investigations have all been performed in linear cascades.

Test performed by Sauer et al. (2000) with a leading edge bulb in a low speed cascade wind tunnel showed incredible secondary loss reductions of approximately 50% compared to the baseline case. The function of the bulb was to intensify the SS-leg of the HS vortex in order to have it interact, by its opposite rotational direction, with the PV such that it would move it away from the SS.

Zess and Thole (2002) tested nine leading edge fillet geometries by CFD simulations in order to determine an optimal design. This resulted in a fillet that was one boundary layer thickness in height and two boundary layers thickness in length. With this fillet, experimental measurements were performed resulting in an elimination of the HS vortex and a delay in the development of the PV. This gave great reductions in turbulent kinetic energy levels and streamwise vorticity levels, which are both large contributors to aerodynamic losses.

In an investigation performed by Becz et al. (2004) comparing a small bulb, a large bulb and a fillet (Figure 2.8) showed that only the fillet gave overall loss reductions (7%). However, they pointed out that the geometries affected the loss in different regions of the span and that additional bulb geometries should be tested. They also speculated that the best geometry would be a combination of the two.

The studies mentioned above have made their pressure measurements upstream and downstream of the blade passage where they are more easily measured. But to get a proper view of the development of the vortices, measurements have to be taken from within the passage. Mahmood and Acharya (2007) have tested two fillets (one with a linear profile and one with a parabolic profile) with measurements taken at four axial positions in the passage and compared them to each other and the baseline
Background

case. Both fillets had a reducing effect on the passage vortex and reduced the pressure loss coefficient across the passage. Generally the linear profile showed best results.

Most experimental investigations deal with aerodynamic losses as opposed to the effects of heat transfer performance. One exception is a study performed by Goldstein et al. (2006). They evaluated the effect of a fillet on endwall mass (heat) transfer. They observed that the fillet eliminated the HS vortex and that the PV was delayed. However, near the TE, the PV had gained as much strength as with the baseline case. They also observed that the filleted case showed higher mass transfer regions near the LE on suction and pressure surfaces. This was explained by the fact that the fillet seemed to intensify the corner vortices at the LE.

Shih and Lin (2002) performed CFD simulations with two different fillet geometries, with and without inlet swirl. The results were compared to baseline case based on surface heat transfer and aerodynamic losses. They found that both swirl and fillets reduced the surface heat transfer by more than 10% on the airfoil and 30% on the endwall. As a measure of aerodynamic loss, the change in stagnation pressure between nozzle inlet and exit was used, and reductions of more than 40% observed.

Another CFD investigation was made by Lethander et al. (2003) with the objective to optimize the shape of a vane leading edge fillet in order to maximize the thermal benefit of the fillet application. The results indicated that a large fillet was the most advantageous, and that the profit was mainly influenced by the height of the fillet.

2.6 Computational fluid dynamics

Computational Fluid Dynamics (CFD) is a computer-based tool that has been used from the 1960s onwards for simulating systems involving fluid flow, heat transfer and other related physical processes (Versteeg 2007). Since the rapid improvement in computing power, powerful graphics and interactive 3D manipulation capabilities in the 1990s, CFD has become an established industrial design tool within a wide range of fields.

“The purpose of computing is insight not numbers” 4 (cited from Versteeg 2007)

CFD is a very powerful tool, which can give valuable and improved understanding of the behavior of a system. It is important to realize that CFD is not a substitute for experimentation but rather an additional tool that should be used on a comparative basis and not for quantitative predictions of real-life systems. Results produced with CFD should be analyzed critically and verified and validated with experimental results continuously.

When used properly, CFD has some great benefits. Even though the investment cost of a CFD-capability is not small, compared to high-quality experimental facilities it is very time- and cost-effective. Another advantage is that CFD codes can produce extremely large volumes of results to practically no added expense. This makes it

---

very suitable for performing parametric studies. Additionally, CFD offers the ability to
study systems where controlled experiments are difficult or impossible to perform.

**2.6.1 Mathematics of CFD**

The most common solution technique used in CFD code is known as *The Finite
Volume Method*. The region of interest is divided into very small control volumes that
together form a grid-structure or a mesh and the user will specify appropriate fluid
properties and boundary conditions in order to start the solution process.

The state of a three-dimensional fluid flow can be fully described in every point by the
velocity (three components), density, temperature and pressure. These six quantities
can be determined from the so called Navier-Stokes equations. These partial
differential equations represent mathematical statements of the conservation laws of
physics and consist of one mass equation, one energy equation and three
momentum equations (Versteeg 2007). However, turbulent flows span a wide range
of turbulent length and time scales, which requires extremely refined meshes and
computer capabilities in order to be solved. For this reason, turbulence models have
been developed and are used for most practical engineering purposes. In this study
the Shear Stress Transport (SST) turbulence model was chosen, which is a two-
equation model. This gives a total of seven equations to be solved.

In the solution process the equations are discretized and solved iteratively for each
control volume. As a result, all variables have an approximated value at nodes inside
each control volume and a full picture of the behavior of the flow is obtained.

**2.6.2 Limitations of CFD**

The assessment of uncertainty is of great importance when determining the accuracy
and level of confidence in any results. In the context of CFD-modeling, the concepts
of *error* and *uncertainty* are widely used and defined accordingly (Versteeg 2007):

*Errors* are recognizable deficiencies in a CFD model that are not caused by lack of
knowledge. They can be divided into the following categories:

- Numerical errors
- Coding errors
- User errors

*Uncertainty* is a potential deficiency in a CFD model that is caused by lack of
knowledge.

*Errors*

Coding errors are mistakes, or “bugs”, in the software and user errors arise from
incorrect use of the software. These human errors can be minimized by substantial
training and experience of the personnel. Numerical errors, on the other hand, are
inevitable in any numerical method.

Convergence errors occur in iterative processes and are identified as residuals. The
normalized global residual is a measure of the difference between the final solution
and the computed solution after a certain number of iterations. In practice, the user
Background

will specify this tolerance and the solution process will finish when the residuals for all equations have reached their preset maximum value. In this project the value was set to $10^{-4}$ for all equations.

Discretization errors occur as approximated distributions of properties between nodal points are used in the solution process. For this reason, decreasing the distance between nodes (refining the mesh) is the main tool for improving the accuracy of a simulation. Having a fine mesh is of particular importance close to solid surfaces where large gradients of flow properties occur. Typically the grid/mesh is refined in stages until no difference can be detected in the results. The results are then called “mesh independent”.

Uncertainty

Uncertainty associated to CFD modeling has two major sources: The ones that are linked to assumptions and simplifications and the ones that are caused by limited accuracy of submodels.

At the start of a CFD project it is common practice to consider whether it is possible to apply any simplifications. Some commonly used simplifications involve treating the flow as:

- Steady vs. transient
- Two-dimensional vs. three-dimensional
- Incompressible vs. compressible
- Adiabatic vs. heat transfer across the boundaries

Complex flow phenomena, such as turbulence and combustion, are embedded in CFD code by submodels. These are derived from high quality measurements and contain empirical adjustable constants which can only capture the type of flow that was used to calibrate their values. Furthermore, the complex flow may involve new and unexpected physical processes that the submodel does not incorporate. On the other hand, the submodel might be a good representation of the flow, but require some adjustments of the constants. In conclusion, the application of submodels can bring uncertainty in a CFD result and the appropriateness of the chosen submodel for the flow to be studied is of great importance.

The absolutely best way to test uncertainties is by validating the models by comparison to experimental results.
3 CFD MODELING

All CFD packages include three main elements: a pre-processor, a solver and a post-processor. The pre-processing stage is the most important and time-consuming one and involves:

- Defining the geometry
- Mesh generation
- Defining the physics of the model

In this project, geometry and mesh generation were done in ANSYS ICEM. The mesh was exported to ANSYS CFX-pre where the physics of the model were defined. ANSYS CFX also contains the solver and the post-processor where results are visualized.

Three models have been created for the purpose of investigation:

- The cascade model (with fillet), based on original CAD design of the ASC
- Ideal model (with fillet)
- Ideal model (baseline)

3.1 Geometry

Before a mesh can be created, a geometric solid defining the computational domain is required. A CAD file of the ASC test rig was imported to ANSYS ICEM (Figure 3.1). In the first step, parts that are not included in the region of interest were removed. The second step involved repairing the geometry, making sure there were no gaps or missing surfaces that would cause problems in the mesh.

![Figure 3.1: Radial section view of CAD model](image-url)
CFD modeling

Additionally, for the cascade model, modifications were made in the downstream part of the cascade to avoid convergence problems due to recirculation. The modifications are illustrated in Figure 3.2 (a) and (b). The ideal/periodic models of one passage were created by building periodic boundary walls on each side of NGV0, seen in Figure 3.2 (c).

Figure 3.2: (a) Cascade model before change (b) Cascade model after change (c) Ideal model
3.2 Mesh

In this project unstructured hexahedral meshes have been produced. The first step in the meshing procedure is referred to as blocking. Upon initialization, one block is created that includes the whole geometry. This block is then split into many blocks and the block topology is generated directly on the underlying CAD geometry. Figure 3.3 shows the block topology for the cascade model, which consists of 180 blocks. The block edges and vertices are then associated to geometry curves and points to capture important features of the geometry.

![Figure 3.3: Block topology of cascade model](image)

The second step is to produce a pre-mesh. This is done automatically after defining the pre-mesh parameters for every edge, such as number of nodes and bunching law.

The quality and refinement of a mesh is of great importance for the convergence and accuracy of the solution.

A good quality mesh indicates that cells are not skewed excessively, meaning that the internal angles of the cell should be as close to 90° as possible. Three parameters have been checked to quantify the quality:

- Determinant
- Volume
- Minimum angle

The determinant is a measure of the deformation of the cells in a mesh. A value of 1 represents a perfect hexahedral cube, while a value of 0 is a completely inverted cube with a negative volume. Generally, determinant values above 0.3 are accepted by most solvers (ANSYS 1).
CFD modeling

The *volume* check computes the internal volume of each cell. It is very important not to have negative volumes inside the mesh since this will cause convergence difficulties in the solver.

Finally, the *minimum internal angle* of each element is checked for further indications of distorted cells.

In ANCYS ICEM, the quality of a mesh can be improved by so called *smoothing*. The procedure is to first smooth the pre-mesh, convert it to a mesh (unstructured in this case) and to perform second smoothing before the *final mesh* is saved. There are a number of different smoothing methods and through an optimization procedure the best method was chosen with respect to the three parameters defined previously (see APPENDIX A).

In general, the accuracy of the solution is better the larger the number of mesh cells. However, there is always a compromise between refinement level, mesh quality and computer capability. Of particular interest is the BL region near the walls where strong gradients of the flow parameters occur and the viscous effects on the transport processes are large. An important parameter in this context is the dimensionless parameter $y^+$, which is used to check the location of the first node away from the wall. It is defined by equation 3.1 with:

\[ y^+ = \frac{\Delta y}{v} \sqrt{\frac{\tau_w}{\rho}} \]  \hspace{1cm} (3.1)

Figure 3.4 shows a turbulent BL profile, which can be divided into the logarithmic layer and the viscous sublayer. In the viscous sublayer, the flow is almost laminar-like and there is a linear relationship between the velocity and the distance from the wall. In this very thin region ($y^+ < 5$) viscosity plays a dominant role in momentum and heat transfer (Versteeg 2007). The logarithmic layer has its name from the logarithmic relationship between the velocity and the distance to the wall, and here the turbulent effects are more important. It typically ranges from $y^+$ values between 30 and 500 (Versteeg 2007). Between these two layers is the so called buffer layer, where the effects of molecular viscosity and turbulence are of equal importance.
The requirement for mesh refinement near the wall depends on the turbulence model and near-wall flow modeling. The k-ω based Shear Stress Transport (SST) turbulence model has been used in this study, which is recommended for high accuracy BL simulation and known to perform well at predicting flow separation under adverse pressure gradients. To benefit from this model, the BL needs to be resolved with at least 10 nodes (ANSYS 1). This was achieved for boundary layers at hub, shroud and sidewalls. However, the node spacing around the vanes had to be increased in order to achieve acceptable mesh quality in these regions.

Wall functions are used to model flow near walls when the mesh is not fine enough to resolve the very thin viscous sublayer. They are based on empirical formulas and relate the logarithmic nature of the velocity profile to compute flow parameters throughout the BL without the need to have it fully resolved. The wall function in CFX is used as a default for k-ε based turbulence models, which are known not to perform well in the regions very close to the wall with low Reynolds numbers (ANSYS 1). For k-ω based turbulence models, the automatic near-wall treatment is used. It has the benefit of automatically switching between a wall function to a low Reynolds number near wall formulation as the mesh is refined (ANSYS 1).

Finally, a grid-dependency study was made for all three cases, where the results from continuously refined meshes were compared based on two parameters; loading and Mach number (APPENDIX B). When the results for the different meshes were stable, the mesh with the smallest averaged y’ value was chosen as the final mesh. For the cascade model this resulted in a mesh with 4 729 060 nodes and for the periodic cases 1 186 972 nodes. The surface meshes for the cascade case and the periodic cases can be seen in Figure 3.5 and 3.6 respectively.
Figure 3.5: Surface mesh of cascade model

Figure 3.6: Surface mesh of ideal models
CFD modeling

3.3 Physics of the model

CFD simulations have been performed with the same inlet and outlet boundary conditions as for the experiment. However, one simplification was made. The total pressure at the inlet was set to be uniform, which means that the upstream BL thickness for the CDF calculations is smaller than for the measurements. In Chapter 2.5 it was mentioned that the inlet conditions for the experiments were varied based on the used turbulence grid. In the upstream plane, 55.7% $c_{ax,hub}$ upstream of the LE, this resulted in BL thicknesses ranging between 8% to 16% of the inlet span (Saha 2012). For the CFD calculations the BL thickness in the upstream plane is around 5% of the inlet span, both at hub and tip endwall as seen in Figure 3.7.

![Figure 3.7: Inlet total pressure profile 55.7% $c_{ax,hub}$ upstream of the LE](image)

The mass flow through the test rig was 2.5 kg/s. The periodic case corresponds to one passage in the ASC, which in total has four passages. However, the massflow for the periodic case is not exactly one fourth (0.625 kg/s) of the total massflow. Through a trial error approach it was found that the massflow in the periodic case was 0.615 kg/s in order to match results from the cascade.

The flow was modeled as compressible flow using “Air Ideal Gas” as working fluid and “Total Energy” as heat transfer model.

The solid boundaries (walls, hub, shroud, blades and fillets) were modeled as adiabatic with no-slip condition.

In Table 3.1, the general settings are summarized.
## Analysis Type
- **Steady state**

## Turbulence Model
- **SST**

## Wall Function
- **Automatic**

## Reference Pressure
- **0kPa**

## Advection Scheme
- **High resolution**

## Timescale Control
- **Auto timescale**

### Convergence Criteria:
- **Residual type**: RMS
- **Residual target**: 1E-4

### Inlet:
- **Total pressure**: 170kPa
- **Total temperature**: 303K
- **Turbulence intensity**: Medium (5%)

### Outlet:
- **Massflow rate Cascade**: 2.5kg/s
- **Massflow rate Periodic**: 0.615kg/s

### Wall Boundaries:
- **Mass and momentum**: No slip wall
- **Wall roughness**: Smooth wall
- **Heat transfer**: Adiabatic

### Table 3.1: General settings used for CFD models
4 AERODYNAMIC INVESTIGATION OF FLOW FIELD QUALITY

Data for the flow field analysis of the experiment\(^5\) have been extracted from the planes in Figure 4.1 and from vane surface lines at 25%, 50% and 75% span. For comparison of CFD and experimental results, data is extracted from these same locations. The inlet measuring plane, Section 1, is located at 55.7% \(c_{ax, hub}\) upstream from the NGV leading edge and downstream data is collected at Section 2, 107.1% \(c_{ax, hub}\). The inlet of the cascade is located at Section 0.

The isentropic Mach number is calculated by equation 4.1 with:

\[
P_1 = \text{upstream total pressure} \\
P_s = \text{static pressure} \\
K = 1.4
\]

\[
M_{iso} = \sqrt{\frac{2}{K-1} \left( \frac{p_1}{p_s} \right)^{\frac{K-1}{K}} - 1}
\] (4.1)

The reference operating point is \(M_{iso3} = 0.9\). Corresponding values for CFD calculations were derived by using the mass averaged total pressure at Section 1 and the static pressures at hub endwall of Section 3 (Table 4.1).

When the annular shape of the models needs to be considered, it is found convenient to use a cylindrical coordinate system. This is illustrated in Figure 4.2 with the inlet plane of the cascade.

<table>
<thead>
<tr>
<th>Model</th>
<th>(M_{iso3})</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cascade</td>
<td>0.86</td>
</tr>
<tr>
<td>Ideal (with fillet)</td>
<td>0.88</td>
</tr>
<tr>
<td>Ideal (baseline)</td>
<td>0.89</td>
</tr>
</tbody>
</table>

Table 4.1: \(M_{iso3}\) at hub endwall

\(^5\) This investigation has been based on a previous experimental study performed by Saha (2012), which is described in Chapter 2.5. Therefore, when referred to experimental data or results, the reference is: (Saha 2012).
Aerodynamic investigation of flow field quality

Figure 4.1: Location of data extraction points for comparison with experiment

Figure 4.2: Inlet plane of cascade: y- and z- coordinates vs. r- and Φ- coordinates
4.1 Load distribution

The turning of the flow through a vane/blade passage sets up a pressure gradient across the passage from the PS to the SS. Thus, across each blade there is a pressure difference generating a lift force that, in the case of rotor blades, is translated into power of a rotating shaft. In this study the loading is displayed as the isentropic Mach number (equation 4.1) over the normalized surface length ($x/c_{ax}$), where the enclosed area is a measure of the vane loading.

Figure 4.3(a) shows the load distribution for all CFD models and the experimental results at 25% span. As can be expected, the fillet does not affect the flow in this region and therefore there is no noticeable difference between baseline and filleted case in the load distribution. Consequently, the effect of the fillet is not noticeable at 50% and 75% span either (see APPENDIX C). Moreover, the agreement with experimental results is very good.

The velocity distribution around a vane is directly related to the pressure distribution on the vane surface, giving information about potential flow separation, shock waves and profile losses. At the PS there are two distinct regions of acceleration: one just after the LE and the other one from about 0.5 of $c_{ax}$, marked by “1” in Figure 4.3 (a) and (b). This delayed acceleration increases the load on the thicker front part of the vane which is good from a load point of view. Along the SS the flow initially increases, but when it encounters the adverse pressure gradient, marked by “2” in Figure 4.3 (a) and (b), starts to decline towards the TE. This may result in a weak oblique shock wave and additional profile losses. To conclude, there is no sign of suction-surface boundary layer separation since this would imply regions of approximately constant static pressure (Saha 2012).
Aerodynamic investigation of flow field quality

As can be seen in Figure 4.4 (a), the geometrical difference between the baseline vane and filleted vane appears below approximately 13% span. The upstream extension of the fillet at the hub is about 30% \( c_{ax, hub} \) and the SS extension is more pronounced than the PS extension. Figure 4.3 (a) and (b) illustrate the extension of the fillet at 10% and 5% span. At 10% span the extension of the fillet on the PS is around 2.5% \( c_{ax} \) and 10% \( c_{ax} \) on the SS. At 5% span the fillet extends 10% \( c_{ax} \) on the PS and 25% \( c_{ax} \) on the SS. In other words, the two blades are geometrically identical to the right of the blue lines marked in the graphs.
Figure 4.4: (a) Extensions of fillet: (b) 10% span (c) 5% span
Aerodynamic investigation of flow field quality

The blade loading at 10% and 5% span is presented in Figures 4.5 (a) and (b) respectively. To the left hand side of the blue lines, the vanes are geometrically different due to the presence of the fillet. In order to see the downstream effect of the fillet, where the vanes are geometrically identical, one needs to look to the right of the blue lines. Looking to the right of the blue lines in both figures, it can be concluded that on the PS the fillet has no effect on the load distribution. On the other hand, on the SS there is a difference between the two cases that lasts up to around 0.4 $c_{ax}$. The load, or the pressure difference, is smaller for the filleted case, which might indicate a weaker endwall crossflow.

In a previous study by Benner et al. (2003), the formation of the HS vortex was shown to depend strongly on the load distribution on the front part of a vane, which has an effect on the pressure field upstream of the LE where the HS vortex is formed. They compared two vanes with different LE geometries and concluded that a more front-loaded vane resulted in stronger secondary flow and consequently larger secondary losses.

When looking at the overall load distribution for the two investigated vanes at 5% span (Figure 4.5 b), the baseline case shows a more front-loaded profile. According to the previously mentioned study (Benner 2003), this indicates that the fillet has a positive effect on reducing the formation of the HS vortex just upstream of the LE.

![Graph showing load distribution at 10% span](image-url)
Figure 4.5: Blade loading: (a) 10% span (b) 5% span
4.2 Flow angles

The exit flow angles together with the radial flow angles form the structure of the flow. In order to take into account the annular shape of the passage, the velocity components were calculated in a cylindrical coordinate system (see APPENDIX D).

In the passage, the flow accelerates and turns. The turning of the flow is expressed by the exit flow angle, $\alpha$, and a smaller angle means higher turning. This angle is set by the shape of the vane and will approximately be equal to the average effective exit angle at the TE, which for this particular NGV is around 16.5° (Saha 2012). In reality, there is always a deviation between the average effective exit angle and the exit flow angle due to for instance; different BL thicknesses on PS and SS, radial flow and secondary flow.

In the mainstream flow, the exit flow angle is relatively close to the average effective exit angle and the largest deviations occur close to the endwall regions. In Figure 4.6, one can see that the two cases show a very similar trend above the secondary flow region close to the hub, where variations are expected to occur due to the fillet. Figure 4.7 illustrates the effect of the fillet close to the hub endwall. The filleted case shows a slightly higher turning compared to the baseline case below around 7% span. The same trend can be seen in the experimental results (Figure 4.8).

Figure 4.6: Pitch-averaged exit flow angle
Aerodynamic investigation of flow field quality

Figure 4.7: Pitch-averaged exit flow angle in secondary flow region

The CFD results show the same trend as the experimental results over the entire span as well, see Figure 4.9. The largest deviations occur in the endwall regions which are dominated by secondary flow phenomena and known to be difficult to capture in CFD. In the mainstream, the discrepancy is between 1° - 2°, which is considered to be within good limits (Utriainen 2012). Additionally, the experimental uncertainty is ± 0.6°.

Figure 4.8: Pitch-averaged exit flow angle in secondary flow region compared to experiment
Figure 4.9: Pitch-averaged exit flow angle comparison with experiment

Figure 4.10 shows the pitch-averaged radial flow angle, $\beta$, over the entire span for the two cases. In agreement with the exit flow angle, small deviations only occur in the hub region where the fillet affects the flow. The negative sign indicates that the flow is downward-directed. This is most probably an effect of the tip endwall contouring of the vane, which is a design feature to reduce secondary flow (see Chapter 2.3). Figure 4.11, illustrates the region of downwash just downstream of the TE by velocity vectors, overlaid with gridlines distributed over the span and the axial distance.
Aerodynamic investigation of flow field quality

**Figure 4.10:** Pitch-averaged radial flow angle

**Figure 4.11:** Meridional view: velocity vectors with gridlines
Vorticity is a measure of the rotation of a fluid element as it moves in the flow field and is an important parameter for secondary flow analysis. The axial component of vorticity ($\omega_x$) is defined according to the following expression:

$$\omega_x = \frac{\partial v_z}{\partial y} - \frac{\partial v_y}{\partial z}$$ (4.2)

As discussed in previous sections, the primary source of the complex vortex flows developing in a turbine passage is known as the HS vortex. The effect of the fillet on the formation of this vortex and its associated legs can be visualized through velocity streamline plots and axial vorticity contours.

The propagation of different vortex structures through one passage, starting at 0% $c_{ax,hub}$ moving to the downstream plane at 107.1% $c_{ax,hub}$, can be followed in APPENDIX E. Through this procedure the different vortices could be identified and labeled in agreement with acknowledged secondary flow models (Chapter 2.1).

Finally, the vortex structures in the downstream plane can be compared for both baseline and filleted cases, including experimental results.

4.3.1 Formation of horseshoe vortex

The formation of the HS vortex upstream of the LE can be identified by the separation lines and the saddle point close to the hub endwall. The region between the separation line and the SS represents the SS-leg of the HS vortex and the region between the separation line and the PS represents the PS-leg of the HS vortex (Acharya 2006). There are some distinct differences in the streamline distributions for the baseline and filleted cases (Figures 4.12 and 4.14). For the baseline case, the separation lines are clearly visible, and it is evident that the flow separates before the LE. The two legs of the HS vortex are clearly seen in the leading edge plane at 0% $c_{ax,hub}$, where the PS-leg is counterclockwise rotating and the SS-leg is clockwise rotating, Figure 4.13.

![Figure 4.12: Velocity streamlines baseline case: (a) Hub and blade (b) blade-to-blade cut (hub)](image-url)
When looking at the filleted case, the flow does not separate before the LE which is evidenced by the absence of the saddle point and separation lines close to the hub just before the fillet (see figure 4.14 b). The streamlines are forced over the fillet, as seen in figure 4.14 (a), and as a consequence, the formation of the HS vortex occurs on the actual fillet. This is also seen in the vorticity contour plots. At the leading edge plane of the filleted case, Figure 4.15, there is no sign of vorticity. Nevertheless, moving downstream over the fillet to the same plane as the LE plane of the baseline case, the two legs of the HS vortex are clearly visible (Figure 4.16). From this it can be concluded that the fillet affects the formation of the HS vortex. However, the overall influence on the development of secondary flow through the passage is best realized when looking at the downstream plane.
4.3.2 Vortex distribution in downstream plane

The vortices are identified and labeled in APPENDIX E, where it is also shown that the fillet has no significant effect on the vortex distribution and intensity above 25% span. Therefore, this section will focus on the vortex structures below 25% span.

The experimental results revealed that the fillet shifted the position of the vortices in the region close to the hub endwall. For the baseline case, the vortices were spread more circumferentially, and for the filleted case, the spanwise extension was slightly higher. Similar trends can be seen in the CFD results.

In Figures 4.17 to 4.20, the positive circumferential direction is marked by the dashed arrow. It is clear from both Figure 4.17 and 4.20, that the circumferential distribution of vorticity is greater for the baseline case. The induced negative vortex is stronger and the PS-leg of the HS vortex is not even present in the filleted case. It has most probably combined with the endwall crossflow to form a more intense PV. From Figures 4.19 and 4.20 it can also be concluded that the filleted case has a more intense CV and a weaker SS-leg of the HS vortex. The magnitudes of the induced positive vortices are nearly the same.
Aerodynamic investigation of flow field quality

The spanwise extension is best realized by comparing the position of the SS-leg HS vortex in relation to the 10% span line in Figures 4.17 and 4.18. The filleted case shows a somewhat higher shift in the spanwise direction.

**Figure 4.17:** Vorticity contour plot downstream plane 107.1% of c_{ax,hub}: Baseline case

**Figure 4.18:** Vorticity contour plot downstream plane 107.1% of c_{ax,hub}: Filleted case
Aerodynamic investigation of flow field quality

Figure 4.19: Axial vorticity at 10% span

Figure 4.20: Axial vorticity at 5% span
Finally, the positions of the vortices coincide well when comparing CFD and experimental results, Figure 4.21.

![Vorticity distribution in downstream plane for CFD and experiment](image)

**Figure 4.21**: Vorticity distribution in downstream plane for CFD and experiment

### 4.4 Aerodynamic loss

As for previously presented results, the differences between the filleted case and baseline case appear close to the hub endwall where the fillet has a direct effect on the flow. Secondary losses in the tip region show the same behavior for the two cases. Also the profile losses remain unaffected by the fillet. Therefore, this section will deal with the secondary losses close to the hub endwall. To see the losses for the whole span, for both CFD and experiment, the reader is referred to APPENDIX F.

The kinetic energy loss is calculated according to equation 4.3 with:

- $P_{2s} =$ downstream static pressure ($107.1\% c_{ax, hub}$)
- $P_2 =$ downstream total pressure
- $P_1 =$ upstream total pressure (here approximated by inlet total pressure)
- $\kappa = 1.4$

$$
\zeta_{kin} = \left( \frac{P_{2s}}{P_2} \right)^{\kappa - 1} - \left( \frac{P_{2s}}{P_1} \right)^{\kappa - 1} \left( \frac{P_2}{P_1} \right)^{\kappa - 1} - \left( \frac{P_2}{P_1} \right)^{\kappa - 1} \right)
$$  \hspace{1cm} (4.3)
It was concluded in the previous section, 4.3.2, that the fillet shifted the positions of the vortices in the hub endwall region. Consequently, this means that the position of secondary losses follow this same distribution. The loss distribution in the downstream plane can be visualized by the normalized total pressure contour plot as seen in Figure 4.22. For the baseline case it is clearly visible that the losses have a greater circumferential spreading, that in turn are caused by the PS-leg of the HS vortex and the induced negative vortex (see Figure 4.20). Furthermore, the loss core is more pronounced for the filleted case, which has higher losses than the baseline case between approximately 15% and 7% span. This can also be seen in figure 4.23. Below this region, there is a shift where the baseline case has higher losses than the filleted case. Eventually very close to the hub endwall the differences between the two cases even out. So to conclude, the fillet shifts the position of losses but the overall difference in total secondary losses is minimal. The same conclusion was drawn from the experimental investigation which shows similar trends as the CFD results, see Figure 4.24. Worth noting here is that in the experiments it was not possible to take measurements all the way down to the hub endwall, which is why experimental data is missing below approximately 6% span.

There is a spanwise shift of the loss core when comparing CFD and experimental results, see Figure 4.24. This is most probably due the fact that the simulations were performed with an inlet condition of constant total pressure. This means that the upstream development of the BL is not as far progressed in the CFD case, resulting in a thinner BL (see Chapter 3.3). The spanwise shift could also be noticed in the exit flow angle distribution (Chapter 4.2).

![Figure 4.22: Normalized total pressure contour in downstream plane](image)
Aerodynamic investigation of flow field quality

**Figure 4.23:** Mass-averaged kinetic energy loss close to hub endwall

**Figure 4.24:** Mass-averaged kinetic energy loss close to hub endwall for CFD and experimental results
5 CONCLUSIONS AND FUTURE WORK

A numerical study and aerodynamic investigation for a leading edge filleted vane and a baseline vane has been performed. Results have been presented on a number of flow parameters describing flow structure and aerodynamic losses. The focus has foremost been to evaluate the effect of the fillet on the secondary flow development. Additionally, computational predictions have been compared to experimental results.

Conclusions

• All results indicate that the fillet affects the flow structure in regions close to the hub endwall. It shifts the position of vortices and loss core, however, the overall effect on reducing secondary losses downstream of the passage is insignificant.

• This investigation differs from previous investigations mainly in two aspects. Firstly, the baseline vane in this study has a typical fillet over the full hub and tip profile, including tip endwall contouring. These are all attributes designed to optimize the performance of the vane on reducing secondary flow. In other investigations the evaluation of the fillet has been based on comparison to a simple baseline case having absolutely no fillet, and therefore no supplementary features for reducing secondary flow. Secondly, the present investigation has been performed in an ASC with relevant pressure gradients, in contrast to previous studies performed in linear cascades. These differences might explain why previous studies have obtained loss reductions that were not found in the present investigation.

• The load distribution indicates that the filleted vane reduces the endwall crossflow and has a positive effect on reducing the formation of the HS vortex just upstream of the LE. Additionally, agreement with experimental results is very good.

• The filleted vane has slightly higher turning than the baseline vane below 7% span. CFD and experimental results show the same trend for the exit flow angles. The largest deviations occur close to hub and tip endwall. In the mainstream, the discrepancy is between 1°- 2°.

• The radial flow angle indicates a similar downwash just downstream of the TE for both baseline and filleted cases. This is most probably an effect of the contoured tip endwall of the investigated NGV.

• The fillet effects the formation of the HS vortex upstream of the leading edge, in agreement with conclusions drawn from the load distribution. Instead of forming upstream of the LE, it forms further downstream on the fillet. In the downstream plane, it can be concluded that for the baseline case, the vortices are spread
Conclusions and future work

more circumferentially, and for the filleted case, the spanwise extension is slightly higher. Finally, the positions of the vortices coincide well when comparing CFD to experimental results.

- The vorticity distribution can also be reflected in the pressure loss distribution. However, the overall secondary loss for the two cases is nearly the same. The same conclusions were drawn from the experimental investigation and the discrepancy between CFD and experimental results for kinetic energy loss is around 1% for the entire span.

- A spanwise shift can be detected when comparing CFD and experimental results for the exit flow angle and the kinetic energy loss. This is most likely caused by the different boundary layer thicknesses in simulations and experiments. In the CFD simulations a constant total pressure has been used as an inlet boundary condition. This results in a boundary layer of about half the thickness of those accomplished in the experiments.

Recommendations for future work

- From the experimental investigation it was concluded that for this configuration, the impact of the upstream field was more relevant concerning secondary losses than the use of a fillet. For this reason, a motivating continuation to the present work would be to incorporate different inlet boundary conditions to the numerical simulations. Interesting parameters to vary are boundary layer thickness and turbulence intensity.

- In order to evaluate the design of the fillet in comparison to other investigations, it would be meaningful to test it on a simple vane on a model representing a linear cascade. If efficiency improvements are gained, it can be concluded that they are only valid under those simplified conditions. On the other hand, if no effects are found, it indicates that other fillet geometries should be tested.

- The present investigation indicates that the typical fillet around the hub and tip profile and the tip endwall contouring has a stronger impact on reducing secondary losses than the employed fillet. This motivates testing other fillet geometries for the same configuration to see if further increased efficiency can be accomplished exclusively attributed to the fillet.
6 REFERENCES


Saha, R. "Aerodynamic Investigation of a High Pressure Turbine Vane with Leading Edge Contouring at Endwall in a Transonic Annular Sector Cascade".
References


APPENDIX A: MESH QUALITY

The settings that were used to smooth the pre-mesh and the mesh are presented together with the values before and after smoothing. Here, the values are taken from the periodic filleted case, but very similar values were obtained with the other models.

![Figure 7.1: Settings for smoothing pre-mesh](image)

![Figure 7.2: Pre-mesh: determinant values before smooth](image)

![Figure 7.3: Pre-mesh: determinant values after smooth](image)
APPENDIX A: Mesh quality

Figure 7.4: Pre-mesh: volume values before smooth

Figure 7.5: Pre-mesh: volume values after smooth

Figure 7.6: Pre-mesh: minimum angle values before smooth

Figure 7.7: Pre-mesh: minimum angle values after smooth
APPENDIX A: Mesh quality

Figure 7.8: Settings for smoothing mesh

Figure 7.9: Mesh: determinant values before smooth

Figure 7.10: Mesh: determinant values after smooth
APPENDIX A: Mesh quality

Figure 7.11: Mesh: volume values before smooth

Figure 7.12: Mesh: volume values after smooth

Figure 7.13: Mesh: minimum angle values before smooth

Figure 7.14: Mesh: minimum angle values after smooth
APPENDIX B: Mesh independence

8 APPENDIX B: MESH INDEPENDENCE

Figure 8.1: Baseline case: Loading

Figure 8.2: Baseline case: $M_{\text{iso}}$ at hub endwall
APPENDIX B: Mesh independence

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Nr of nodes</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Hub</td>
</tr>
<tr>
<td>a</td>
<td>1 151 540</td>
<td>179.444</td>
</tr>
<tr>
<td>b</td>
<td>1 186 972</td>
<td>170.782</td>
</tr>
<tr>
<td>c</td>
<td>1 204 688</td>
<td>179.287</td>
</tr>
<tr>
<td>d</td>
<td>1 186 972</td>
<td>137.116</td>
</tr>
</tbody>
</table>

Table 8.1: Baseline case: Mesh density and averaged y+ value

![Figure 8.3: Filleted case: Loading](image-url)

Figure 8.3: Filleted case: Loading
APPENDIX B: Mesh independence

Figure 8.4: Filleted case: $M_{iso2}$ at hub endwall

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Nr of nodes</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Hub</td>
</tr>
<tr>
<td>a</td>
<td>1 056 000</td>
<td>256.218</td>
</tr>
<tr>
<td>b</td>
<td>1 151 540</td>
<td>181.691</td>
</tr>
<tr>
<td>c</td>
<td>1 186 972</td>
<td>149.692</td>
</tr>
<tr>
<td>d</td>
<td>1 204 688</td>
<td>141.915</td>
</tr>
<tr>
<td>e</td>
<td>1 228 800</td>
<td>167.168</td>
</tr>
<tr>
<td>f</td>
<td>1 257 836</td>
<td>149.517</td>
</tr>
<tr>
<td>g</td>
<td>1 186 972</td>
<td>138.856</td>
</tr>
</tbody>
</table>

Table 8.2: Filleted case: Mesh density and averaged $y^*$ value
APPENDIX B: Mesh independence

Figure 8.5: Cascade case: Loading

Figure 8.6: Filleted case: $M_{iso2}$ at hub endwall

Table 8.3: Cascade case: Mesh density and averaged $y^+$ value
APPENDIX C: LOAD DISTRIBUTION

**Figure 9.1**: Load distribution 50% span

**Figure 9.2**: Load distribution 75% span
APPENDIX C: Load distribution

**Figure 9.3:** CFD: Load 75% span

![Image of a graph showing load distribution at 75% span with labels for Cascade fillet, Periodic fillet, and Periodic baseline.]

**Figure 9.4:** CFD: Load 50% span

![Image of a graph showing load distribution at 50% span with labels for Cascade fillet, Periodic fillet, and Periodic baseline.]

APPENDIX C: Load distribution

**Figure 9.5: CFD: Load 25% span**

**Figure 9.6: CFD: Load 13% span**
Figure 10.1: Velocity components for calculating exit flow angle and radial flow angle (adapted from ANSYS 2)

\[
\cos \alpha = \frac{v_\Phi}{v_{b2b}} \quad (10.1)
\]

\[
\tan \beta = \frac{v_{sp}}{v_{b2b}} \quad (10.2)
\]
The axial vorticity is presented through the passage between NGV0 and NGV-1 for the cascade case. Starting at 0% $c_{ax, hub}$, moving to the downstream plane at 107.1% $c_{ax, hub}$. The passage is cut into planes and seen from the downstream point of view according to figure 11.1. The clockwise rotating negative vortices are marked in a blue colour and the counterclockwise rotating positive vortices are marked by a red colour.

**Figure 11.1**: (a) Axial vorticity scale (b) Downstream viewpoint of extraction-plane within passage

The two legs of the HS vortex have formed on the PS and SS of the fillet.

**Figure 11.2**: Axial vorticity at 0% $c_{ax, hub}$
APPENDIX E: Vorticity

**Figure 11.3: Axial vorticity at 20% $c_{ax, hub}$**

The PS-leg of the HS vortex moves towards the SS and combines with the endwall crossflow to form the PV.

**Figure 11.4: Axial vorticity at 60% $c_{ax, hub}$**

A CV has appeared on the PS. The SS-leg of the HS vortex is pushed upwards by the emerging PV/PS-leg of the HS vortex.
**Figure 11.5: Axial vorticity at 80% \( c_{ax,\text{hub}} \)**

At 80% \( c_{ax,\text{hub}} \) there is a split of the large positive vortex known as the PV. In this study, the two resulting vortices are named differently: The one that positions itself under the SS-leg is called the PV and the other one is referred to as the PS-leg of the HS vortex.

**Figure 11.6: Axial vorticity at 90% \( c_{ax,\text{hub}} \)**
At 95% c_{ax, hub} the negative TE shed vortex is formed as a result of the BL separation just downstream of the TE, and is a big source of profile losses. The positive vortex between the TE shed vortex and the CV appears just downstream of the TE on the PS of the blade and is probably induced by the surrounding vortices.

**Figure 11.8:** The resulting vortices in the downstream plane at 107.1% c_{ax, hub}
APPENDIX E: Vorticity

**Figure 11.9:** Baseline case: whole downstream plane with spanlines

**Figure 11.10:** Filleted case: whole downstream plane with spanlines
Figure 11.11: Axial vorticity at 25% span

Figure 11.12: Axial vorticity at 50% span
APPENDIX E: Vorticity

**Figure 11.13:** Axial vorticity at 75% span

**Figure 11.14:** Axial vorticity at 90% span
APPENDIX F: AERODYNAMIC LOSSES

The CFD profile losses range between approximately 20% span and 80% span. Below 20% and above 80% the secondary flow influence is apparent and hence these losses are referred to as secondary losses. For the experimental case, due to the spanwise shift described in Chapter 4.4, the region of profile losses is defined from 30% span to 70% span.

The overall (for the entire span) mass-averaged kinetic energy loss is for the baseline case around 4.55% and for the filleted case around 4.62%. Practically this means that the losses can be considered equal and that the fillet has no reducing effect on the secondary loss. The discrepancy between the CFD results and the experimental results is around 1% for the total span.

Figure 12.1: Mass-averaged kinetic energy loss distribution CFD
APPENDIX F: Aerodynamic losses

Figure 12.2: Mass-averaged kinetic energy loss distribution experiment

Figure 12.3: Mass-averaged kinetic energy loss distribution: Comparison CFD and experiment