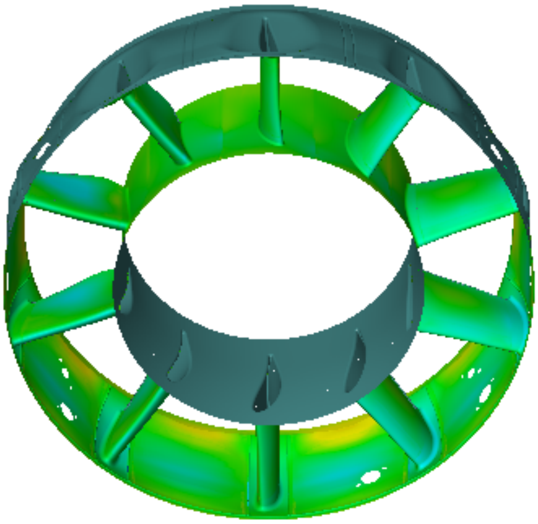
CHALMERS



# Improvement of Aerodynamic Performance Predictions for a Component with Manufacturing Features

Master of Science Thesis in the Master Degree Programme Product Development

Oscar Linde
Department of Product and Production Development
*Division of Product Development*
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden, 2013

Improvement of Aerodynamic Performance Predictions for a Component with Manufacturing Features
OSCAR LINDE

Department of Product and Production Development
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone + 46 (0)31-772 1000

Cover:
Figure illustrates the geometric deviations for a manufactured turbine rear structure (Figure 13 in the report).

Chalmers Reproservice
Göteborg, Sweden, 2013

## Abstract

GKN Aerospace Sweden AB in Trollhättan, Sweden, develops and manufactures a range of components for aircraft and rocket engines. One of these components is the Turbine Rear Structure (TRS) used in civil aircraft jet engines. Aerodynamic evaluations on manufactured structures are essential to confirm that requirements for aerodynamic performance are fulfilled. One important estimate of aerodynamic performance is the pressure loss over the TRS. In order to evaluate the performance of a manufactured TRS with CFD (Computational Fluid Dynamics), GKN uses a white light scanning technique to build up a three dimensional model. This master's thesis studied how manufacturing features, such as welds and weld preparations, affect the aerodynamic performance. A method was developed to use the scanned data of the manufactured TRS to evaluate its sectors individually in CFD. Performance for manufactured sectors was compared to the performance of models from CAD (Computer Aided Design). Then, it is possible to detect were the losses appear. Manufacturing features perpendicular to the flow direction were identified to be the features most affecting pressure loss. These could explain about 60% of the higher losses due to manufacturing. Other manufacturing features could explain the most of the remaining part. Hence, a minor remaining part was a loss due to geometric deviations from nominal definition. This thesis also studied how the effects of manufacturing features could be reduced by changes in design or manufacturing process. The improvements were mainly ways of reducing the height of the manufacturing features, either during the welding process or as an after treatment.

## Acknowledgements

I would like to show my gratitude to my supervisor Linda Ström at GKN Aerospace Sweden AB and to Professor Hans Johannesson at Chalmers University of Technology. I would like to thank Bernhard Gustafsson at GKN for all help with the software program ANSA. Finally, I also want to thank all colleges, internship students and master's thesis students at GKN who have supported me during the time of my thesis.

## Nomenclature

| | | |
|---|---|---|
| $P_t$ | [Pa] | Total Pressure |

## Abbreviations

| | |
|---|---|
| CAD | Computer Aided Design |
| CFD | Computational Fluid Dynamics |
| GAS | GKN Aerospace Sweden |
| GOM | Gesellschaft für Optische Messtechnik (Optical Measuring Techniques) |
| GKN | Guest, Keen and Nettlefolds |
| k-ω SST | The k-ω SST turbulence model |
| MFG | Manufacturing |
| OEM | Original Equipment Manufacturer |
| rk- ε | The realizable k-ε turbulence model |
| TEC | Turbine Exhaust Case |
| TRF | Turbine Rear Frame |
| TRS | Turbine Rear Structure |

# Contents

1 Introduction 1

1.1 GKN Aerospace Sweden AB 1

1.2 Civil Jet Engines 1

1.3 Turbine Rear Structure 2

1.4 Manufacturing Features on a Turbine Rear Structure 2

2 Thesis Background and Purpose 4

2.1 Problem Background 4

2.2 Purpose 4

2.3 Research Questions 4

2.4 Objectives 5

2.5 Delimitations 5

3 Frame of Reference 6

3.1 Software 6

3.2 CFD Analyses 6

3.2.1 Turbulence Models 7

3.2.2 Guide Vane Aerodynamics 7

3.2.3 Total Pressure Loss 8

3.3 Design and Manufacturing Improvement Methods 9

4 Approach 10

4.1 Design Practice Method Development 10

4.2 CFD Analyses 11

4.2.1 Geometries 11

4.2.2 Simulation Running 13

4.2.3 Boundary Conditions and Flow Conditions 13

4.2.4 Mesh Resolution and Turbulence Models Dependency 13

4.3 Potential Design and Manufacturing Improvements 14

5 Results 15

5.1 Design Practice Method Development 15

5.1.1 Summary of Design Practice Method 15

5.1.2 Buffer Zone Effects 16

5.2 Analyses of Nominal and Real Sectors 17

5.2.1 Geometric Deviations 18

5.2.2 Boundary Layer Flow 20

5.2.3 Total Pressure Loss 20

5.2.4 Mesh Resolution and Turbulence Model Dependency 25

5.3 Analyses of Repaired Real Sectors ........ 26
5.3.1 Manufacturing Features at Upstream Shroud ........ 26
5.3.2 Manufacturing Features at Downstream Shroud ........ 27
5.3.3 Manufacturing Features on Vane – Welds on Vane ........ 29
5.4 Identification of Total Pressure Losses ........ 30
5.5 Design and Manufacturing Improvement ........ 31
5.5.1 Root Causes for High Total Pressure Loss ........ 31
5.5.2 Potential Solutions of Root Causes ........ 31
5.5.3 Evaluation of Solutions ........ 32
6 Discussion ........ 34
6.1 Design Practice Method ........ 34
6.2 CFD Analyses ........ 34
6.3 Design and Manufacturing Improvements ........ 35
7 Conclusions ........ 36
8 Future Work ........ 36
9 References ........

# 1 Introduction

This introducing chapter begins with a presentation of the company GKN Aerospace Sweden AB. The principles of civil jet engines will be described and the Turbine Rear Structure is introduced. It is the effect on aerodynamic performance from manufacturing features on this component that will be studied in this thesis.

## 1.1 GKN Aerospace Sweden AB

This thesis work is executed at GKN Aerospace Sweden AB in Trollhättan, Sweden. The company will throughout this report also be referred to as GAS (GKN Aerospace Sweden). GAS is a part of the worldwide GKN Aerospace, which is the aerospace operation of GKN plc. GKN Aerospace focuses on three major product areas - aero structures, engine products and transparencies. The business has significant participation on most major civil and military programs. GKN Aerospace is also a major supplier of integrated composite structures. GAS develops and manufactures components for aircraft, gas turbines and rocket engines with high technology content in cooperation with the world's leading OEMs (Original Equipment Manufacturers), such as General Electric, Rolls Royce and Pratt & Whitney. GAS also offers a range of services, including sales of spare parts for aircraft engines and aircraft, sales and leasing of aircraft engines, as well as overhaul and repair of aircraft engines and industrial gas turbines. GAS was until 2012 called Volvo Aero, which was the year Volvo AB sold Volvo Aero to GKN Aerospace. [1]

## 1.2 Civil Jet Engines

Jet engines all use the same principles and can also be called gas turbines. An overview of a typical jet engine is show in Figure 1.

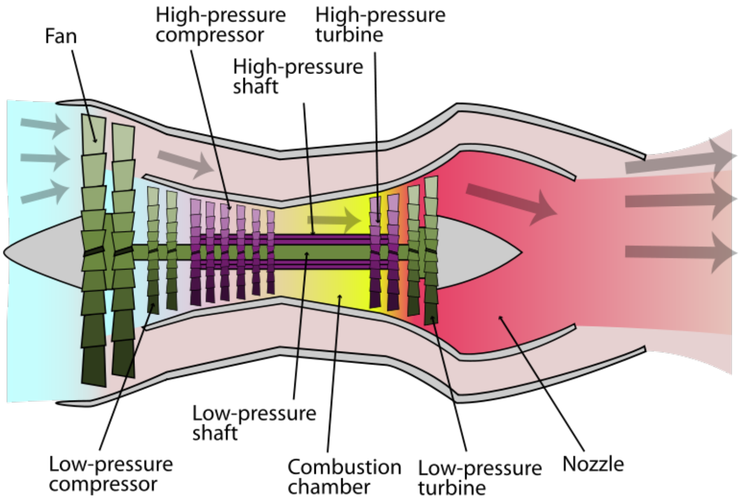


**Figure 1. Schematic diagram illustrating the operation of a low-bypass turbofan engine. Source: http://en.wikipedia.org/wiki/File:Turbofan_operation_lbp.svg.**

The principle is that air is sucked in through the inlet, where some air will pass only the fan on its way through the engine, and some air through the core of the engine. In the core, air is compressed, first by a low-pressure compressor and then by a high-pressure compressor. This increases the pressure and the energy potential of the air. In the combustion chamber, fuel is injected and ignited by electric sparks. The burning gases expand, increasing the airflow speed through the engine. After the combustor there are one high-pressure and one low-pressure turbine running the shafts for fan and compressors. Eventually, the hot air exits the nozzle with a higher speed than at the inlet. This is what creates thrust. The nozzle could be followed by a mixer, which mixes the hot and cold air helping the engine to become quieter.

Modern engines used for civil air transport are usually of a kind called turbofans, which have a high-bypass ratio. This means that a major part of all incoming air at inlet only passes the fan and not the core of the engine. The air flowing around the engine core is called bypassed air. The cool air from the bypass makes the engine quieter and adds thrust. Therefore, turbofans work as gas turbines, but the fan also has the function of a propeller. [2]

## 1.3 Turbine Rear Structure

The Turbine Rear Structure, also called TRS, is a component developed and manufactured by GAS for commercial aircraft. Depending on OEM, the TRS can also be denoted TRF (Turbine Rear Frame) or TEC (Turbine Exhaust Case). The TRS is located in the rear of the engine downstream the low-pressure turbine, see Figure 2. A TRS has several functions. One function is to compose one of the frames carrying the engine and is mounted to the wing. Another function is to enable oil to enter and exit shafts and bearings. A third and most important function for this thesis is the aerodynamic function of diverting flow from the low-pressure turbine. The air exiting the low-pressure turbine is slightly swirled and needs to be diverted to axial direction. A number of aerodynamically shaped guide vanes create this diversion. The guide vanes have a shape similar to a typical airfoil.

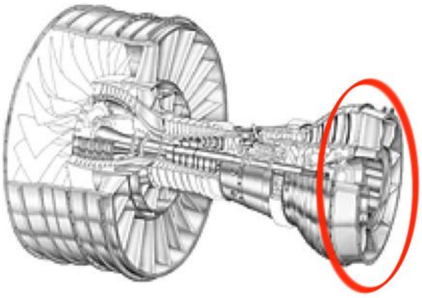


**Figure 2. Illustration of a jet engine. The TRS is circumscribed. (Source: GKN Aerospace)**

GAS offers a lightweight technology by fabrication, meaning that several sections are put together to create one component. The fabrication manufacturing technique leads to manufacturing features that can affect the technical functions, such as the aerodynamic performance and thus the specific fuel consumption.

## 1.4 Manufacturing Features on a Turbine Rear Structure

The manufacturing by fabrication of a TRS creates manufacturing features that deviate from nominal design. Deviations from nominal design do generally affect the aerodynamic performance negatively. These features are mainly welds and weld preparations and can both be circular features around the TRS, but also axially in the flow direction. The height of a weld increases with thicker materials and differs by weld technique and also along the weld itself. A cross-section of a weld is showed in Figure 3.

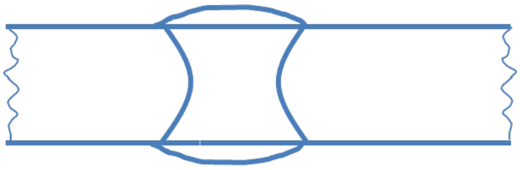


**Figure 3. Cross-section of a weld.**

Weld preparations can be used to avoid mismatches when welding. This is conducted by having one or both of the joining materials with an exaggerated thickness, which is then machined to a flush match (Figure 4). Depending on the mismatch of the materials at start, the height of the remaining weld preparation can be higher or lower. After this machining procedure, the weld joining of the two parts is made through the contact surface.

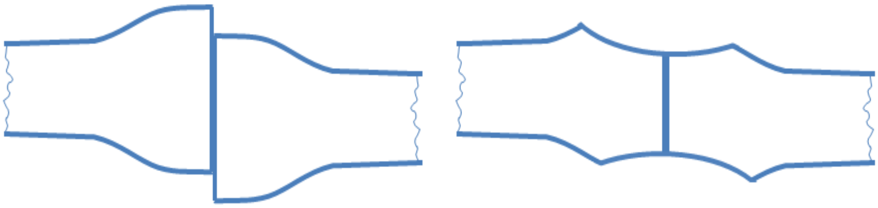


**Figure 4. A mismatch of materials before welding is avoided with weld preparations. To the left are both joining materials of a higher thickness, which to the right are machined to a flush match.**

Throughout the product development process, geometric information is at GAS gathered with 3D scanning techniques in order to verify fulfilment of drawing tolerances and dimensions, which are set to fulfil technical requirements. It is used for evaluation and improvement of the manufacturing process, but is also useful for examination of the dimensions and placements of the manufacturing features. GAS uses a white light scanning technique from here on called GOM (Gesellschaft für Optische Messtechnik), which is the name of the company offering these optical measuring techniques. The technique is based on 3D scanning with two objectives photographing the component and generating a 3D point cloud. From this point cloud, a 3D mesh is built up. This representation of a manufactured component can be used as geometry in Computational Fluid Dynamics (CFD) and its performance can be compared the performance of a nominal geometry.

# 2 Thesis Background and Purpose

This chapter describes the frames of the thesis. It begins with a problem background and purpose. The research questions are formulated, as well as the objectives of the thesis, and also the delimitations are given.

## 2.1 Problem Background

A trend in the aircraft industry is to develop more fuel-efficient aircraft by for example weight reductions of the aircraft and engines. This means new materials and new manufacturing methods. There is no exception for the TRS, where changed manufacturing methods result in new manufacturing features that make the manufactured component deviate from the aerodynamically ideal product. In the development of a new TRS, these features are important to examine and it is necessary to evaluate the aerodynamic performance losses due to them. A common estimate for evaluation is the total pressure loss that is obtained from inlet to outlet of a TRS.

In order to adjust the manufacturing process before serial production, several components are produced. These components are used for component and engine tests. For every new component of these pre-serial TRSs, adjustments can be made to ensure that the geometric deviations meet the requirements during serial production.

There is an interest from GAS's point of view to make it possible to evaluate these aerodynamic effects more in detail to confirm that the requirements for performance losses are fulfilled. At this point, GAS has a recently developed design practice method to evaluate a full 360 degrees TRS representation from GOM scanning in CFD. It is computationally demanding to manoeuvre and simulate a full TRS model, which makes the resolution of surfaces and flow suffer. Hence, GAS has a wish to enable examination of one sector with one guide vane at a time in order to resolve the manufacturing features more than before. With a more resolved model the origins of the pressure losses can hopefully be located and corrected. Except the issue of resolution, sector wise evaluation can also be useful in other aspects.

## 2.2 Purpose

The purpose of this thesis is to improve the understanding of the manufactured component features and their effects on the aerodynamic performance for a TRS. The thesis work should develop a methodology in order to evaluate the manufacturing features more comprehensively, by sector wise evaluation. It is aimed to evaluate the effects of a variety and locations of manufacturing features. This thesis is also supposed to examine how changes in design and manufacturing could improve the performance.

## 2.3 Research Questions

- How can aerodynamic performance of manufactured TRS sectors be evaluated individually in CFD?
- Which manufacturing features are significant for the aerodynamic performance of a TRS?

- To what extent do manufacturing features affect the aerodynamic performance?
- What potential improvements in design and manufacturing are possible to reduce the influence of these manufacturing features?

## 2.4 Objectives

- Development of a design practice method describing the procedure from scanned GOM model to sector wise evaluation in CFD for use in current and later projects at GAS
- Improve methods to investigate real geometry effects
- Define which manufacturing features that are critical for performance
- Reduce uncertainty with aerodynamic performance predictions

## 2.5 Delimitations

This thesis is delimited for validation of TRSs or similar structures. The deviations considered are only geometric deviations. Only aerodynamic aspects are considered, not structural, thermal or other aspects. For this thesis only the total pressure loss due to manufacturing is relevant. Other requirements than total pressure loss, such as separation sensitivity and swirl of the outlet flow, are not in focus. Evaluation will only include studies of the conditions at the aerodynamic design point. The aerodynamic design point represents the conditions at the phase of flight where the TRS is designed for highest efficiency. Other phases of the flight cycle are not considered. Some verification of CFD accuracy will be conducted, but since this is a thesis focusing on product development and improvement and not theoretical CFD analysis, it will not be a major part.

Existing best practices for CFD at GAS are assumed to be accurate enough to give reliable results for its purposes. The design practice method development is delimited to the software offered by GAS.

# 3 Frame of Reference

This chapter will give some theoretical background to increase the understanding of the following chapters. It begins by describing the different software programs that will be used for the design practice method. Some theory about CFD is presented, which is necessary for understanding of the upcoming results. Also, a brief background to the method for design and manufacturing improvements is described.

## 3.1 Software

All software programs used for developing the design practice method and generating CFD results are explained below.

**GOM Inspect**

GOM (white light scanning) is an optic measuring technique with two objectives where a high amount of pictures from different views create a 3D point cloud of an object. From this point cloud, a 3D mesh is generated. This 3D mesh is then compared to a nominal CAD model and all dimensional deviations from nominal geometry are visualized in a range of colours. The deviations are computed in GOM as the normal difference of the GOM surface and the nominal surface.

GOM offers several software programs for different applications. The software for this thesis work is their free edition of GOM Inspect. GOM Inspect is a 3D inspection and mesh processing software for dimensional analysis of scanned geometries. GOM Inspect version 7.5 was used in this thesis for dimensional analyses and export of surface meshes of scanned TRSs.

**ANSA**

ANSA is an advanced multidisciplinary pre-processing tool that provides necessary functionality for full model build-up. ANSA has the functions of treating both surface meshing and volume meshing. For this thesis, ANSA version 14.0.1 was only used for surface meshing and creating real sector surfaces for each sector separately.

**ANSYS ICEM CFD**

This is an advanced meshing software program for various fields of application. For this thesis work, ANSYS ICEM CFD version 14.0 was used for creating volume meshes for later use in CFD simulations.

**ANSYS Fluent**

ANSYS Fluent contains the broad physical modelling capabilities needed to model flow, turbulence, heat transfer etc. The procedure of this software is not part of the developed design practice method, since there exists already a GAS best practice. For this thesis work, version ANSYS Fluent version 14.0 was used for CFD simulations.

## 3.2 CFD Analyses

CFD (Computational Fluid Dynamics) is an advanced method of simulating fluid dynamics. The flow domain is broken into a fine mesh of elements and nodes, which algebraically simulate the basic partial differential equations of flow. CFD analyses are more cost efficient compared to empirical wind tunnel tests. CFD allows engineers to construct a geometry and boundary conditions to simulate a given viscous flow problem. A mesh is created in the software and attempts to compute flow properties at each mesh element. Convergence of the solution is essential for a successful simulation. Turbulent flows are not completely resolved by the full equations of motion. Approximate turbulence models are used to model this turbulent flow. [3]

### 3.2.1 Turbulence Models

Turbulence models are developed for particular geometries and flow conditions. No single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model depends on several factors such as physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. [3]

#### Realizable k-ε Model (rk- ε)

The Realizable k-ε model is a relatively new developed model, originating from standard k-ε models. The term "realizable'' means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. This turbulence model predicts accurately the spreading rate of both planar and round jet flows. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. The realizable k-ε model is proven to be conservative when it comes to total pressure loss predictions, i.e. it tends to predict higher losses than other turbulence models. [4]

One disadvantage with this model is the production of non-physical turbulent viscosities in situations when the computational domain contains both rotating and stationary fluid zones. Also, it is prone to under-predict separation of boundary layers.

#### Shear-Stress Transport (SST) k-ω Model (k-ω SST)

The shear-stress transport (SST) model uses k-ε in the free stream and k-ω in the wall-bounded region. Works well with adverse pressure gradients and separated flow. This model needs fine mesh close to the wall [5]. It is proven to over-predict separation.

#### Transition SST Model (k-ω SST with transition)

k-ω SST with transition is based on a coupling of the SST k-ω transport equations with two other transport equations. k-ω SST tends to result in lower pressure losses. It is possible to examine intermittency, which is an estimate of the time when the boundary layer is laminar or turbulent.

### 3.2.2 Guide Vane Aerodynamics

A guide vane is similar to a typical airfoil regarding part names and properties, see Figure 5. To describe the axial location in a fluid domain, one can generally say upstream and downstream. Upstream is in the direction of inlet and downstream is in the direction of the outlet. The upstream edge of a vane is called leading edge and the downstream one trailing edge. Chord is the length of the vane from leading to trailing edge, and span is the depth of the vane "into the paper" [3]. Wake is the low velocity and low total pressure area after the vane, where flow from suction side and pressure side meets again.

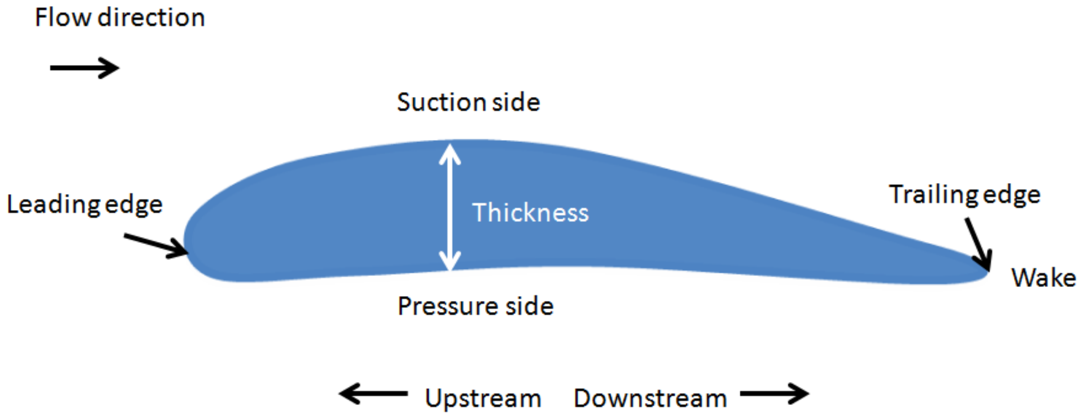


Figure 5. Illustration of names necessary to for this thesis.

The flow velocity is high and the static pressure is low at suction side. At pressure side it is the opposite, low velocity and high pressure. This pressure difference will result in a diversion of the flow that also generates lift.

The flow near a solid surface is called boundary layer. In a boundary layer, flow velocity is zero at the surface and increase from the surface to reach free flow at the top of the boundary layer. Boundary layers can separate from the surface, creating negative flow velocity. Separation is not desirable and it is a major concern for aerodynamic performance.

### 3.2.3 Total Pressure Loss

Total pressure loss is a measurement of aerodynamic performance, which is defined as the pressure difference from inlet to outlet:

$$Pt\ loss = \frac{Pt\ inlet - Pt\ outlet}{Pt\ inlet}$$

In order to find the origins of losses, it is beneficial to determine the pressure loss difference between two cases:

$$Delta\ Pt\ loss = Pt\ loss - Pt\ loss\ reference$$

The loss change from a reference case can be stated as a relative change in per cent:

$$Pt\ loss\ change = \frac{Delta\ Pt\ loss}{Pt\ loss\ reference}$$

## 3.3 Design and Manufacturing Improvement Methods

Information and data for possible improvements was gathered through a brainstorming session with experts. A scoring matrix was used for evaluation of potential improvements of the manufacturing features regarding aerodynamic performance. The matrix is a modified version of an existing matrix at GAS to evaluate the potential of improvements. The overall idea of the matrix is to present the problem and find its root causes. To each root cause, several solutions are suggested. Every solution is evaluated on its benefits and drawbacks by assigning numbers.

- 1=low benefit/drawback
- 3=medium benefit/drawback
- 9=high benefit/drawback

# 4 Approach

This chapter will discuss the approach that was used during the project. It will describe how the design practice method was developed from existing design practice method. The approach for the CFD analyses is also described and it is explained how the simulations were conducted. At the end of the approach chapter, the approach to generate and evaluate potential improvements for better aerodynamic performance is described.

## 4.1 Design Practice Method Development

The initial phase of the thesis work was intended to create a knowledge base about the topic and software. Reading internal GAS documents and also general information found on the Internet. Different alternative approaches to the design practice method development were discussed. It was early decided that this design practice method should be based on the software already used for the similar purposes by GAS. Therefore, the recently developed GAS design practice method for evaluating full 360 degrees TRSs worked as a starting point. The existing design practice method does only consider the part where ANSA is used. This design practice method was developed at GAS in 2013 [6]. This design practice method was tried out to learn about the procedure and what changes that had to be done for the new one.

Individual sector validation needs a strategy for how the simulation domains are defined. For nominal geometries, the interfaces between two sectors are always identical. This allows the simulation domain to be periodic, meaning that flow exiting on periodic surface enters from the other periodic surface. Though, for real geometries these interfaces are never the same, due to geometric deviations. A strategy with “buffer zones” was a solution for this variation (Figure 6). The simulation domain will have one inlet, one outlet and periodic surfaces seen in Figure 8.

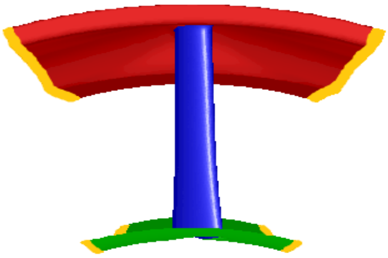


**Figure 6. Buffer zones highlighted in orange.**

It had to be decided what steps should be done in each software program. Corresponding actions were tried in GOM Inspect and ANSA to choose the most appropriate approaches of the two. It is mainly between these two software programs alternative approaches to solve a certain problem can be done differently. For the developed design practice method, a GOM surface was used to create a surface representation in GOM Inspect. Then ANSA was used to improve the mesh quality of the representation.

The knowledge regarding the GOM technique and the software GOM Inspect was gained by attending a one-day course at Cascade in Mölndal, Sweden. Cascade is the provider of the GOM technique to GAS. The course was attended the 15th of March 2013 and increased the understanding of the GOM technique and the usage of GOM Inspect.

## 4.2 CFD Analyses

When the design practice method was developed, it was used to create the geometries for the CFD simulations. Manufactured TRSs were to be compared to nominal CAD geometries. To verify the reliability of the CFD simulations, the influence of the volume mesh resolution and different turbulence models were considered. The TRS is located right downstream the low-pressure turbine. Due to this, the conditions at the low-pressure turbine outlet work as inlet conditions to the TRS.

### 4.2.1 Geometries

In this section the geometry of a TRS and its different sector types will be presented. Scanned manufactured TRSs will also be called "real" geometries later in this report. Figure 7 shows the naming of all sectors for this thesis, viewed in the rear the engine.

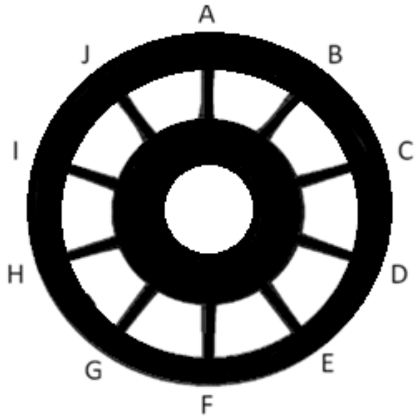


**Figure 7. TRS from rear view with sector names.**

One separate sector and its simulation domain with according names are displayed in Figure 8. The sector continues some distance after the actual TRS outlet in order to avoid influence from the TRS exit plane. In reality, the TRS is only ranging from Inlet to TRS outlet. The interface between low-pressure turbine and TRS inlet needs some simplifications, since there is a play causing air leakage between low-pressure turbine and TRS shroud. The reason for this play is that the low-pressure turbine is rotating and the TRS is fixed.

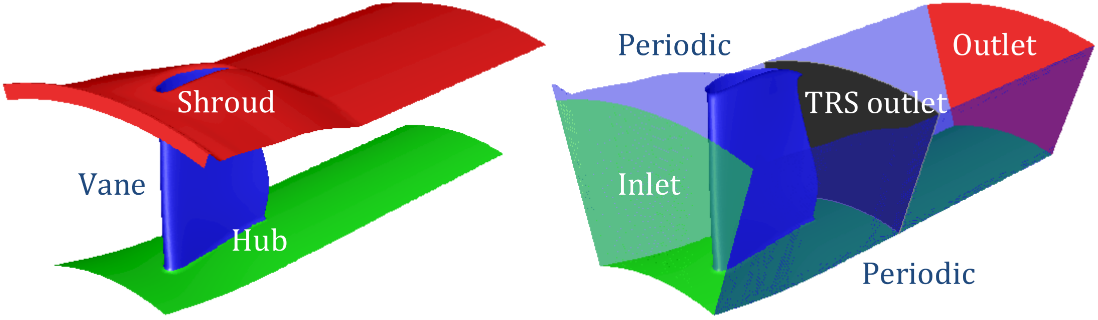


**Figure 8. One sector consists of shroud, vane and hub. The simulation domain consists of one inlet and one outlet. Where the TRS ends there is an evaluation plane, TRS outlet. The sidewalls are periodic surfaces.**

The TRS is composed of three different types of sectors:

- **Mount sectors** are located at the top of the TRS. These sectors are mounted to the wing and carry most of the structural load. Therefore they are manufactured in thicker materials and other manufacturing procedures than regular and tube sectors. The mountings enforce the presence of a bump around the vane at shroud.
- **Tube sectors** contain tubes for oil for lubrication of bearings and do therefore have a higher vane thickness.
- **Regular sectors** have ordinary vanes without any added functions.

The locations of the manufacturing features considered for this TRS are illustrated in Figure 9. The features are into the gas channel. They are not the same for all sectors in a TRS.

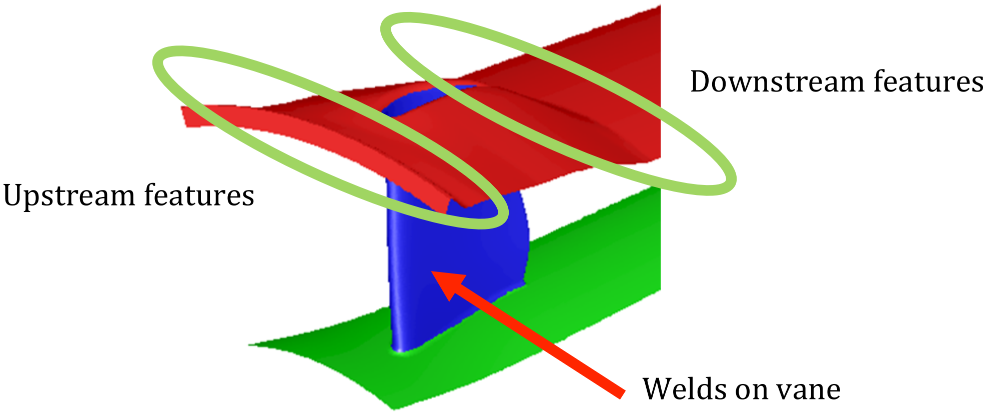


**Figure 9. Illustration of the areas with manufacturing features considered in this thesis.**

### Nominal sectors

In order to evaluate the real geometries, also nominal geometries have to be analysed under the same circumstances. Nominal geometries are models that are created in CAD, both with and without manufacturing (MFG) features. The MFG geometry includes manufacturing features in terms of modelled weld-preparations, but no welds are modelled in these geometries. There is also a difference between “cold” and “hot” model. A cold model has the dimensions of a TRS at room temperature and a hot model has the dimensions of the TRS in service (around 350°C for the aerodynamic design point) [7]. The hot model is slightly bigger due to the expansion of materials. The TRS is designed so that the aerodynamic performance is optimal when it has the hot model dimensions. Despite this, this thesis analyses only cold models, since the manufactured TRSs are scanned at cold conditions. The relative difference in performance between manufactured and nominal model is evaluated.

### Real sectors

Real sectors are the scanned manufactured TRS sectors. These sectors have both welds and weld preparations. The developed design practice method makes it possible to analyse these sectors in CFD separately. There were two scanned manufactured TRSs considered for this thesis. These two components will from now on be called TRS 1 and TRS 2.

**Repaired real sectors**

For evaluation of the magnitude of the pressure loss increase for each manufacturing feature, so-called repaired sectors were created. This means that features being considered to influence the performance were repaired one at a time to determine the effect of each feature. The reparations were made in such a way that the features were changed to represent a surface similar to the nominal surface, without the feature.

Three different manufacturing features were studied more closely; upstream shroud features, downstream shroud features and welds on vane. The reason for choosing these particular features was because these were considered to cover most of the possible manufacturing features that could result in a higher pressure loss.

### 4.2.2 Simulation Running

Volume meshes of the fluid were created in ANSYS ICEM CFD. ANSYS Fluent was used to create the CFD models. After the calculations, several journal files were run to create the required post-processing files and images. Journal files set all constants and settings automatically. For TRS 1 all real sectors were analysed, but for TRS 2 only two of each sector type. The reason for this was that there was more time for analysing TRS 1 than for TRS 2. All sectors were analysed for TRS 1 to evaluate the spread among sectors. Nominal and nominal manufacturing geometries were simulated in order to compare the results to the real geometries.

All cases were simulated with as similar volume meshes as possible to allow comparisons. Only small mesh adjustments had to be made due to geometric deviations in the geometry.

### 4.2.3 Boundary Conditions and Flow Conditions

The TRS is located next downstream to the low-pressure turbine. The outlet conditions from this turbine work as inlet conditions for the TRS. See the GAS document [7] for accurate boundary conditions. The same boundary conditions have been used during the GAS design iterations. Inlet flow angle is around 20 degrees from axial. The axial interface surfaces between all sectors are periodic. This means that air exiting at one periodic surface will enter from the other periodic surface. Approximating the conditions to periodic surfaces is a way of simulating each sector as if it was simulated as a full TRS with only one type of sectors. This is a simplification, since the surrounding sectors have an influence of the conditions on each individual sector. For comparisons under similar circumstances, this is anyway an accurate approach.

### 4.2.4 Mesh Resolution and Turbulence Models Dependency

To evaluate the dependence on the volume mesh resolution, several cases were run. The mesh resolution was specifically changed around the guide vane. One regular real sector from TRS 1 and one regular nominal sector were evaluated with three mesh resolutions, low-, medium- and high-resolution. The medium mesh size was the size used as a standard mesh had around 6.2 million nodes, the less resolved one around 2.2 million nodes and the more resolved one around 18.1 million nodes.

The turbulence model used for most cases was rk-ε. The aim of using different turbulence models was to discover how the results changed depending on turbulence model used. Therefore, the turbulence models k-ω SST and k-ω SST with transition were also simulated. The three turbulence models are proven to have certain characteristics. This was conducted for a regular sector from TRS 1 and a nominal regular sector, both for low-, medium- and high-resolution meshes.

## 4.3 Potential Design and Manufacturing Improvements

The results obtained from the CFD simulations were discussed with a manufacturing leader at GAS in order to get input for the improvement work. Significant manufacturing features and locations for negative impact on aerodynamic performance were discovered in the analyses. Potential solutions for each feature were discussed and evaluated on benefit for performance, cost and risk. The author of this thesis, the author's supervisor and a manufacturing leader were the persons participating in this session. The meeting started with a brief background and results of the CFD analyses from this thesis. As soon as the problems were communicated, the meeting turned into a discussion of how these problems occur, potential solutions and effects of each solution. All results were summarized in a table discussing root cause, solution, benefit on aerodynamic performance, cost and risk of implementation. This was more thoroughly documented in an internal document at GAS. [YY]

# 5 Results

The result section is divided into design practice method development, CFD analyses and potential improvements. The results of the developed design practice method will cover an overview of the developed method. This is followed by the results from the CFD analyses for nominal and real manufactured sectors. These results include geometric deviations of the real TRSs, low velocity boundary layers, total pressure losses and the results from the mesh and turbulence model study. Later, the results from the repaired sectors are presented in order to get the magnitude of losses for certain manufacturing features. There is also an identification of the losses that can be explained by manufacturing features and the ones that cannot. In the end, the results from the design and manufacturing improvements are presented.

## 5.1 Design Practice Method Development

Some results are confidential and therefore is the design practice method summarized, the complete design practice method is kept as an internal document at GAS [8]. Some evaluations of the buffer zone effects were conducted to confirm reliability.

### 5.1.1 Summary of Design Practice Method

The design practice method starts from the point when having a scanned GOM surface mesh in GOM Inspect. Compared to the existing design practice method at GAS more of the surface mesh modifications are executed in GOM Inspect instead of ANSA. Then there are certain steps that have to be conducted for examination of one sector, compared to a full TRS. Below is the developed procedure to evaluate a real TRS sector in CFD.

1. **Reduce GOM mesh**
   For better computational manoeuvrability it is recommended to reduce the scanned surface mesh to only aerodynamic surfaces. These are the surfaces exposed by flow in the gas channel. The reduction is simplified if a CAD model is created with only the surfaces needed. With this model the selection and deletion of mesh is fairly simple. The reduction will also simplify the following filling of the holes in the mesh.
2. **Fill mesh holes**
   The holes in the mesh caused by lack of scanning data have to be filled. There should not be any holes left on the aerodynamic surfaces.
3. **Export as STL format**
   When the mesh is reduced and all holes are filled, the mesh is exported as STL format.
4. **Import mesh into ANSA**
   The STL model from GOM Inspect is imported in ANSA.
5. **Import nominal geometry file into ANSA**
   Merge the STL with a geometry file consisting of nominal domain and split surfaces for all sectors. The geometry file is a ANSA file, where one geometry file per TRS type has to be created. The geometry file can be created either in CAD software or in ANSA. In ANSA this

geometry is meshed to be able to generate the following wrap mesh. This geometry file is the major modification for this design practice method compared to the existing one.

6. **Wrap mesh**
   A wrap mesh is a mesh that is created on the inner walls of a closed cavity. Each sector and its split geometry build these closed cavities. The reason for wrap meshing is to increase the surface element quality. The increase of element quality makes the models manoeuvrable in CFD. This design practice method results in meshes for each sector with buffer zones in between.
7. **Fill gaps to nominal geometry for each sector**
   To create a comparable domain for each sector, the wrapped mesh is filled to domain inlet, domain outlet and periodic surfaces.
8. **Export sectors as CGNS format**
   When all sector domains are filled to geometry surfaces, each sector is exported as CGNS format.
9. **Import surface mesh into ANSYS ICEM CFD**
   The CGNS file is imported as a mesh in ANSYS ICEM CFD. The function "Mesh to Facets" creates surfaces from the surface meshes (Figure 10). Curves and points around the surfaces need to be created.

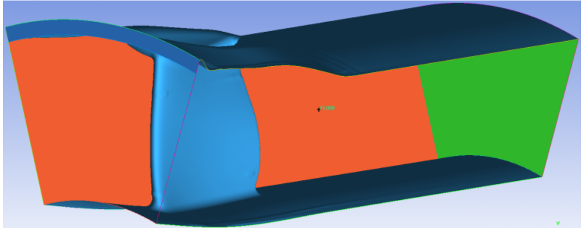


**Figure 10. Surfaces of the simulation domain. Inlet and one periodic surface are hidden.**

10. **Import blocking**
    A blocking file consists of blocks with volume meshes. This blocking file is assumed to exist for a similar domain and only needs to be adjusted to fit the actual geometry. The volume element quality regarding element angles and determinants etc. follows GAS's best practice to obtain reliable results.
11. **Export volume mesh as MSH format**
    When the volume elements fulfil the GAS requirements, the mesh is exported in ANSYS Fluent format.

### 5.1.2 Buffer Zone Effects

There are potential drawbacks of the developed design practice method. Obvious areas that could influence the results are the buffer zones to the periodic surfaces.

The effects of these buffer zones were examined by static pressure and axial shear stress at the actual cut. Figure 11 illustrates that there are some characteristics, which may be caused by buffer zones. Compared to nominal geometries' pressure curve, the real geometries have some small-scale variations at the buffer zones.

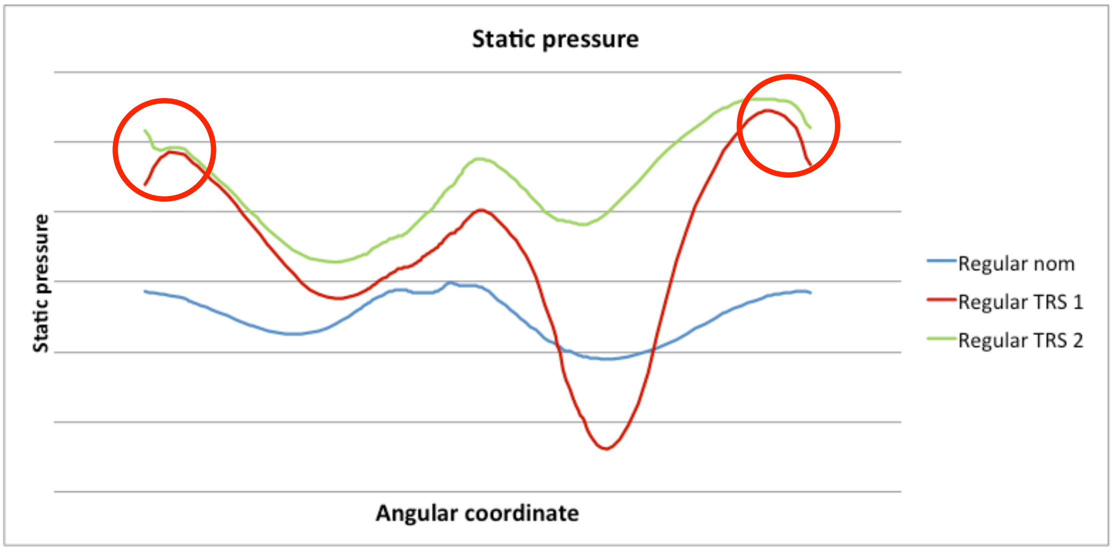


**Figure 11. Static pressure at TRS shroud. At curve ends for TRS 1 and 2, the effects of buffer zones are circumscribed.**

The variations caused by buffer zones are small compared to other variations, and therefore not considered significant.

## 5.2 Analyses of Nominal and Real Sectors

This chapter presents key results from the CFD analysis part. Further information can be found in GAS documents [9]. Note that figures visualizing individual guide vanes will be shown from different views depending on the purpose of each figure. Axes in diagrams are either removed or normalized. Normalization of axial length shows the range from TRS inlet to outlet on a scale from 0 to 1. Total pressure loss normalization also uses the scale from 0 to 1, where 0 represents no pressure loss and 1 represents a certain value, which is the same for all cases to allow comparison.

Axial coordinates are also normalized to 0 for TRS inlet and 1 for TRS outlet. To investigate where the pressure losses of a TRS initiate, it is informative to evaluate the total pressure at several axial locations. Figure 12 shows the ten evaluation planes considered. The data received at each cut is an average value of each cut surface. Figures in this chapter show the pressure side view with the gas flowing from left to right. Inlet is represented by 0 and TRS outlet (x_outlet) as 1.

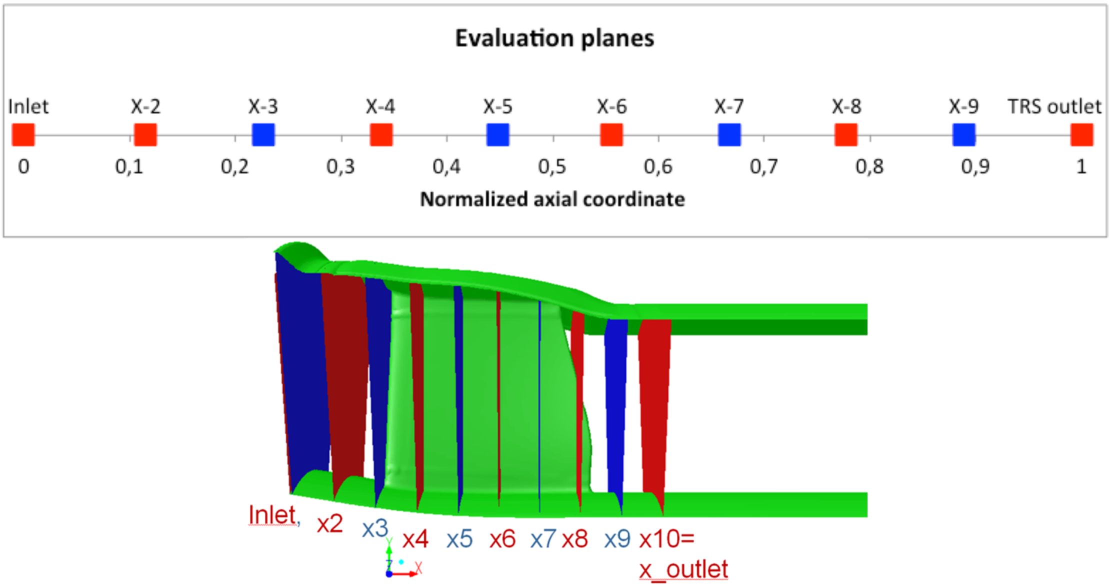


**Figure 12. Evaluation planes for the sectors. TRS outlet and X-outlet are the same plane.**

### 5.2.1 Geometric Deviations

Figure 13 shows an example of how a surface comparison is displayed in GOM Inspect. Green areas represent geometric deviations around 0mm from nominal. Yellow areas represent positive deviations and blue areas negative deviations.

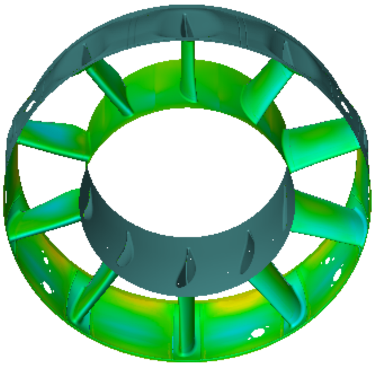


**Figure 13. Front view of the aerodynamic surfaces of a TRS with geometric deviations from nominal surfaces.**

TRS 2 has less deviation on the mount sectors than TRS 1. Common for both TRSs is the positive deviation at downstream shroud on sectors D-H, but they are higher for TRS 2. Most vanes do also have a slightly more chambered shape, i.e. there is a positive deviation on suction side and negative on pressure side.

The manufacturing features (welds and weld preparations) are not visible in the figure above. These are local features and the ones considered in this thesis. Figure 14 is a cross-section of the shroud from Inlet to X-3 on regular sector H. The first bump is a circular weld and the second is a circular weld preparation.

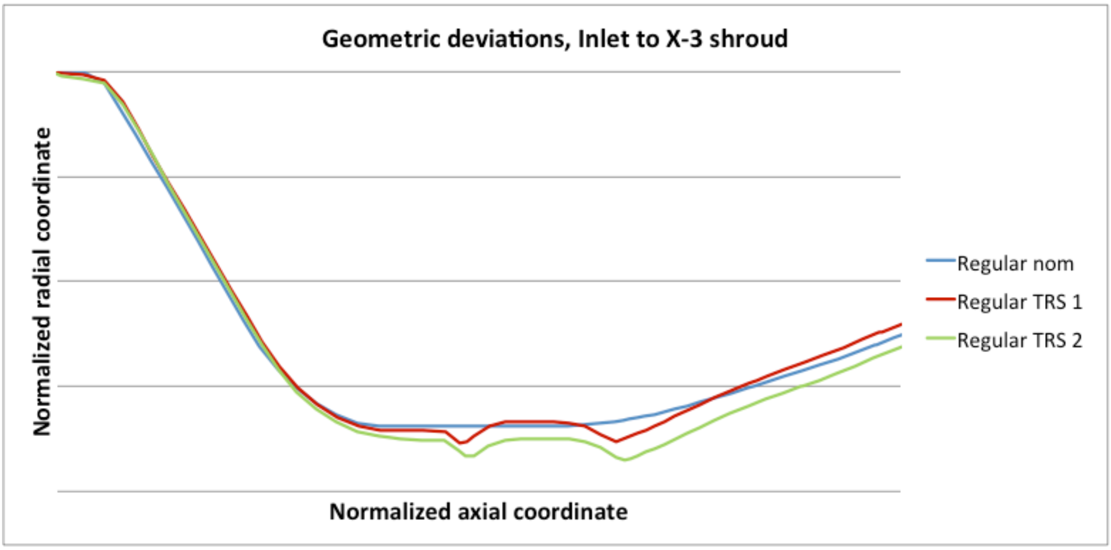


**Figure 14. Cross-section showing the upstream shroud manufacturing features.**

Further downstream, the shroud is positively deviated and there is also a circular weld. Figure 15 shows a cross-section for this area at centre of a regular sector. The circular weld is seen in the figure.

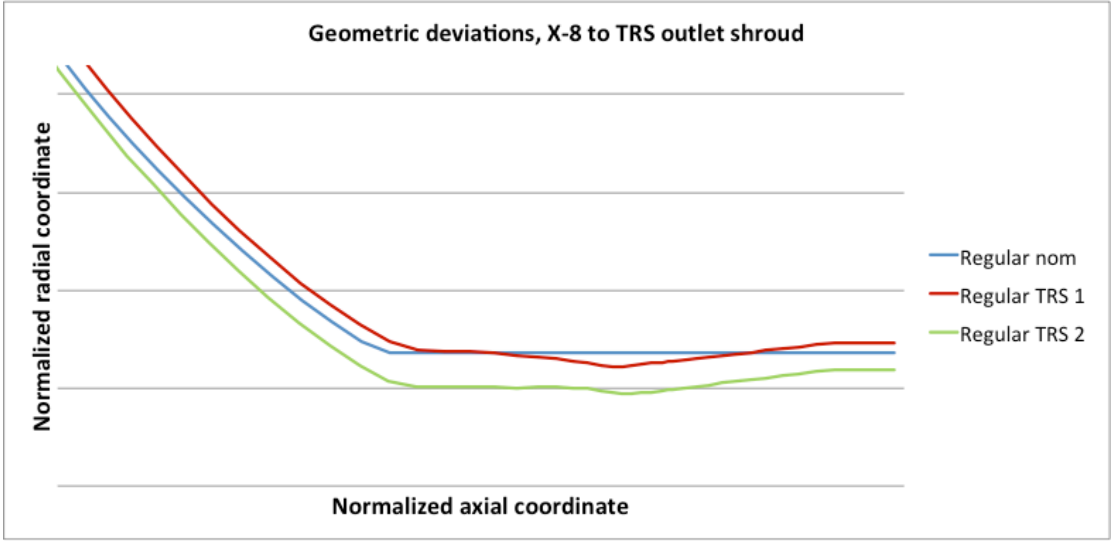


**Figure 15. Cross-section showing the downstream shroud manufacturing features in sector centre.**

Figure 16 shows the same axial location as in Figure 15, but between two vanes.

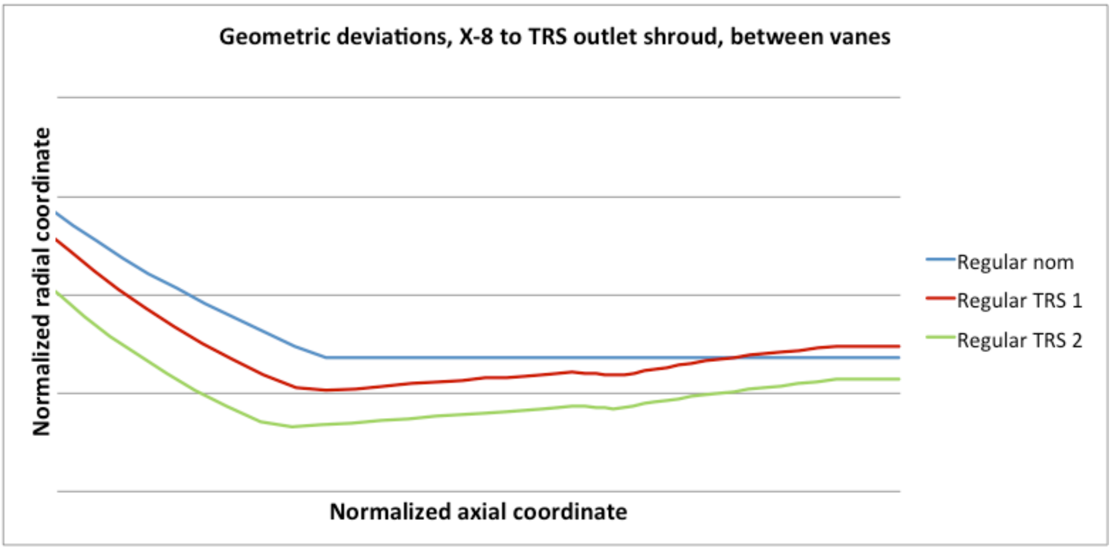


Figure 16. Cross-section showing the downstream shroud manufacturing features, between vanes.

The manufactured welds on vane have such a small height that they cannot be seen in a cross-section of a vane. Vanes have more global displacements than local deviations.

### 5.2.2 Boundary Layer Flow

Figures that show boundary layers with low velocities are informative for discovering where total pressure losses are originated. Figure 17 shows the low velocities in dark grey for a nominal regular sector. The suction side of a guide vane is more probable than the pressure side to have low velocities, which is the reason why the following figures have the suction side view with the gas flowing from right to left. The first low velocity region that is present for all simulated cases occurs because of the pocket next downstream the low-pressure turbine. This is no result from manufacturing features and hence a loss not considered in this thesis.

Figure 17. Low flow velocities for a nominal regular sector. The shroud is hidden for better visibility.

Low velocity regions for nominal geometry with manufacturing features (no welds, only weld-preparations) are slightly different. The boundary layer has low velocities after the circular weld-preparation at shroud.

For real sectors, there are generally low velocity regions after the weld and weld preparation at shroud flange. There are only small differences in low velocity areas between TRS 1 and 2.

### 5.2.3 Total Pressure Loss

Total Pressure loss is an essential measurement for aerodynamic performance. A higher pressure loss means lower performance. All diagrams in this chapter have a normalized pressure loss from 0 to 1, where 1 represents a certain total pressure loss value. This value is the same for all diagrams to allow comparison.

The total pressure loss varies around a TRS. Figure 18 summarizes the results from nominal sector simulations and simulated sectors for TRS 1 and 2. Nominal sectors with manufacturing features have a higher loss compared to nominal due to their weld preparations. Mount sectors do not have as high loss increase as tube and regular sectors. For nominal sectors there are no differences within each sector type, e.g. all regular sector results are the same simulation. It is only for real sectors that all sectors are individual.

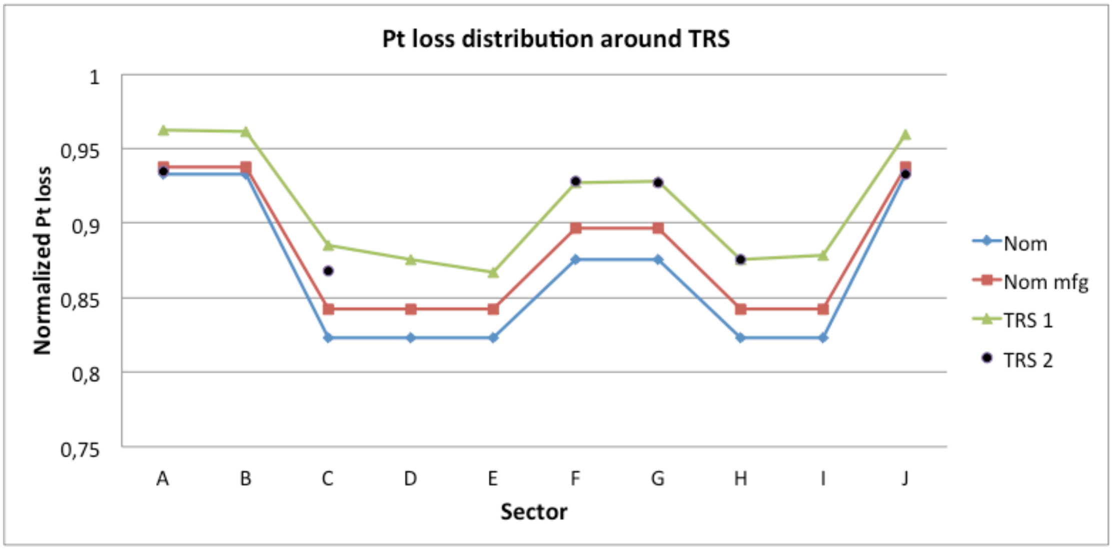


**Figure 18. Pressure loss distribution around TRSs when sectors simulated individually.**

When simulating a full TRS there are effects among the sectors, which do not occur when simulating sector wise. The explanation is basic aerodynamics. Mass flow rather takes a path with low blockage than a path with high blockage. This means that more mass flow will go through the sectors with low-pressure loss (regular sectors) and less through the sectors with high-pressure loss (mount and tube sectors). More mass flow results in more loss and less flow results in more loss, so in reality the graph over pressure loss would be a bit more levelled out.

Figure 19 shows how the pressure loss increases axially through nominal TRS sectors, from inlet to outlet. The end value at TRS outlet is the total value of the total pressure loss. In the following figures is the vane range displayed. “Vane short” and “Vane long” start at the same coordinate, but they end at different coordinates. “Vane short” represents the range, which is the shortest vane range, and “Vane long” represents the longest vane range. Figure 19 shows that losses for mount sectors compared to regular sectors appear where the shroud bump starts, at around vane start. For tube sectors, the losses appear along the vane.

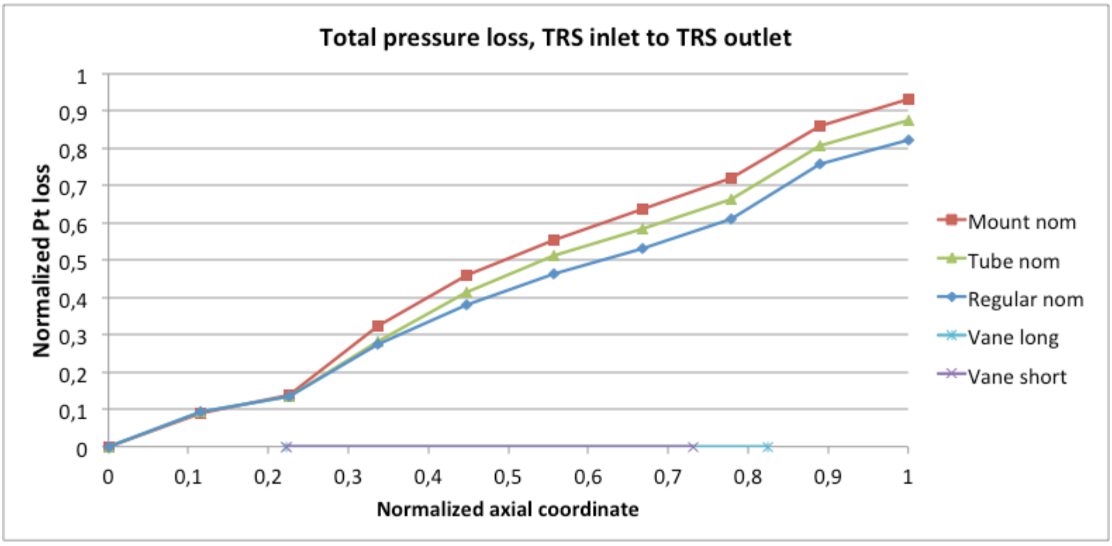


**Figure 19. Curves illustrating the pressure loss build-up from inlet o TRS outlet of nominal mount, tube and regular sectors.**

The trends are similar for manufactured sectors, but generally with slightly higher loss. The nominal regular sector and a corresponding real sector are plotted in Figure 20.

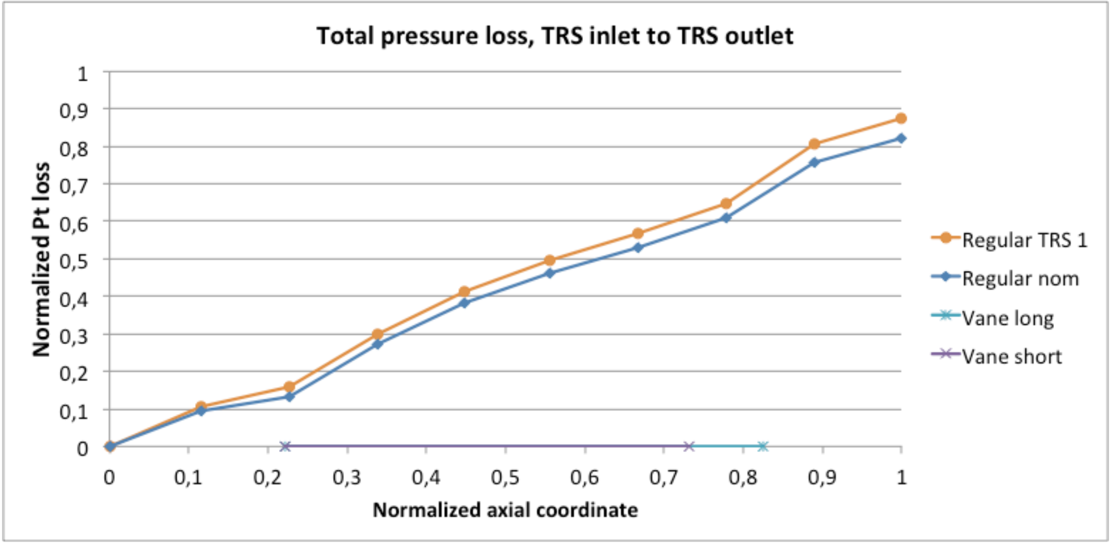


**Figure 20. Curves illustrating the pressure loss increase for a regular sector due to manufacturing.**

It is logically a higher pressure loss for real manufactured sectors, due to manufacturing features and other geometric deviations from nominal geometry. In plots like Figure 20 it is difficult to detect the origins of the pressure loss increase. Therefore, the delta for total pressure loss between nominal and real sectors is a more detailed way of finding where the losses appear.

Results from the nominal sectors work as reference, and hence the losses for a real compared to nominal sector can be identified. If the delta for total pressure loss is positive, the real model has a higher pressure loss than the nominal sector. In other words, the results of a nominal sector would follow the x-axis completely, at zero Pt loss. In Figure 21 to Figure 24 all evaluated sectors for TRS 1 and 2 are represented by delta graphs compared to nominal sectors. At each evaluation plane seen in Figure 12, the difference in total pressure loss is determined (real model loss subtracted with nominal model loss). The nominal manufacturing (with modelled weld-preparations, but no welds) is plotted for all sector types for better understanding. Figure 21 shows the pressure loss delta for TRS 1 mount sectors.

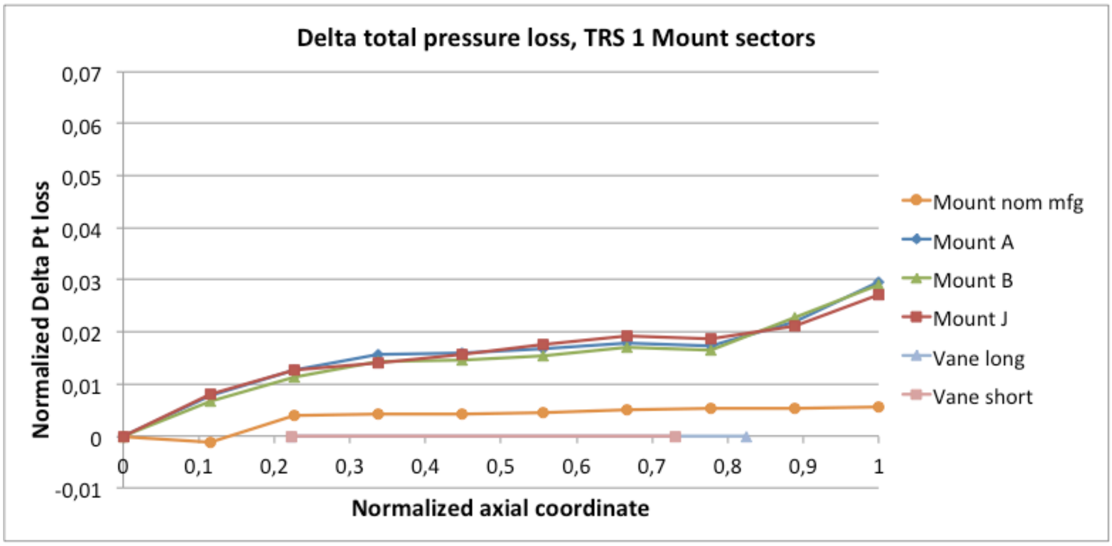


Figure 21. Pressure loss increase from nominal, TRS 1 mount sectors.

Figure 22 shows the pressure loss delta for TRS 1 tube sectors.

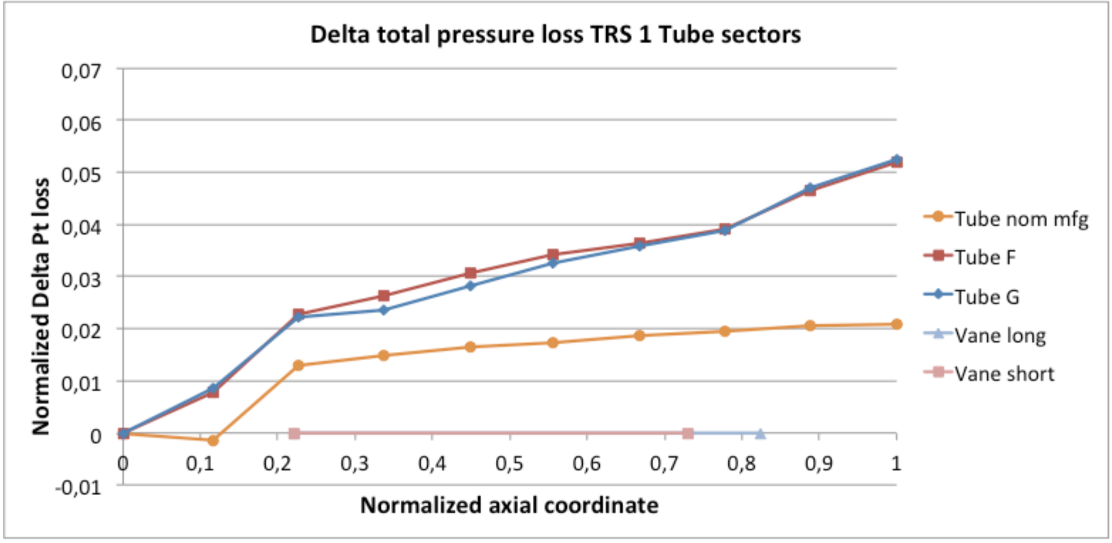


Figure 22. Pressure loss increase from nominal, TRS 1 tube sectors.

Figure 23 shows the regular sectors for TRS 1. The spread among sectors initiates already before the vane. Along the vane are all results fairly parallel, but in the wake there is a various loss increase among the sectors.

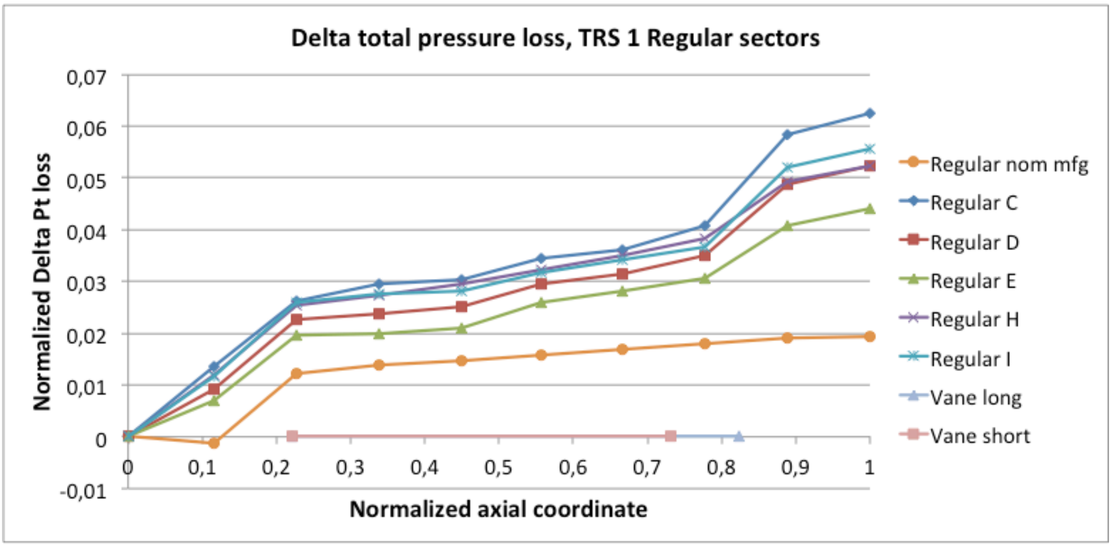


Figure 23. Pressure loss increase from nominal, TRS 1 regular sectors.

In the plots above, the effects of weld and weld-preparations at upstream shroud are consistent for all types. To clarify the interpretation of these plots it is important to understand where the evaluation planes are, showed earlier in Figure 12. The first plane is the TRS inlet, second is after the circular weld and the third plane is after the circular weld-preparation at shroud. This means that the effects of these features can be clearly seen in the plots. These features add a substantial part of the overall total pressure loss increase. It was in Chapter 5.2.2 noted that this area generally had low velocity regions in the boundary layer. Along the vanes, the loss increase is somewhat linear and does not increase with as high rate as upstream the vane. At the vane wake, the losses increase again. Noteworthy is the increase from X-8 to X-9, where the loss increase varies for the sectors. After X-9, all cases are parallel.

The results from TRS 2 indicate similar tendencies as the ones for TRS 1. The dissimilarities are mainly for the mount sectors; compare Figure 24 to Figure 21. Real sectors are actually improving the pressure loss compared to nominal sectors and there is a decrease of delta at the vane wake.

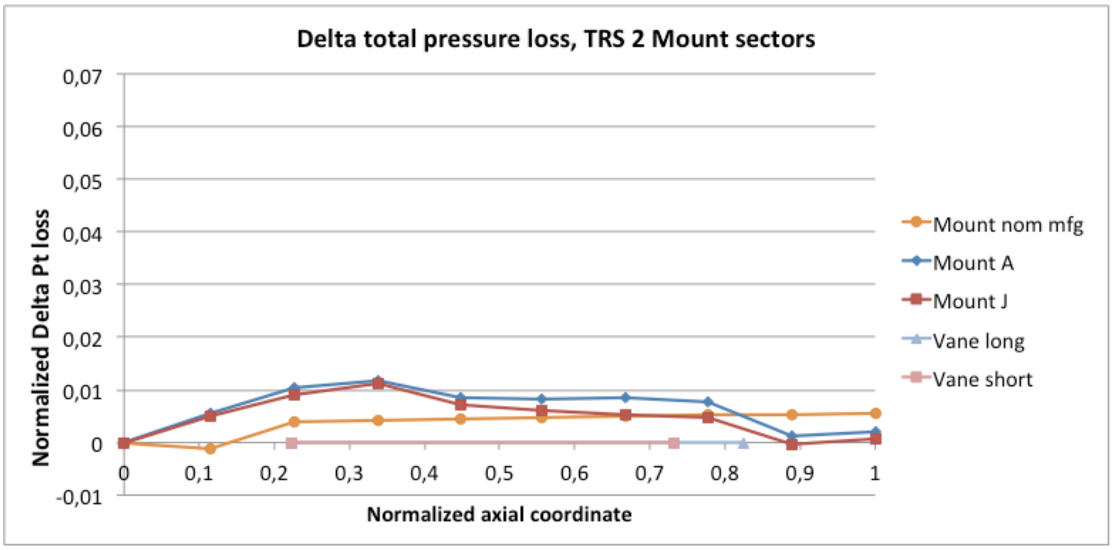


Figure 24. Pressure loss increase from nominal, TRS 2 mount sectors.

### 5.2.4 Mesh Resolution and Turbulence Model Dependency

The effect of various mesh resolutions on pressure loss can be seen in Figure 25. The results are for a real regular sector, but it is similar for a nominal sector. General for all cases is that low-resolution obtains the highest total pressure loss, and the difference between standard- and high-resolution meshes is hardly noticeable. The losses for the low-resolved case originate in the wake. It indicates that the resolution dependency of manufacturing features is not noticeable.

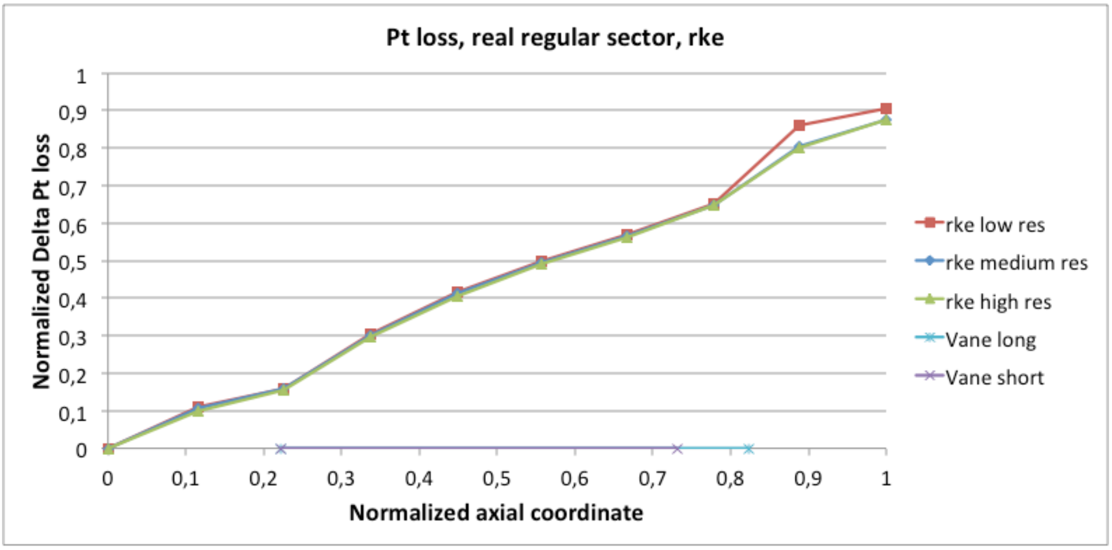


**Figure 25. Pressure loss for real sector with low- medium- and high-resolution of volume mesh.**

Noteworthy is the small inconsistency in mesh resolution on trailing edge.

- Low resolution: 2.2 million nodes
    - Trailing edge: 12 nodes

- Medium resolution: 6.2 million nodes
    - Trailing edge: 18 nodes

- High resolution: 18.1 million nodes
    - Trailing edge: 15 nodes

This inconsistency was made to get a functional mesh. Except this, all other parts of the mesh follow the low- medium- and high-resolution node numbers. Though, Figure 25 shows the importance of the wake resolution. If the trailing edge resolution is low, the wake is less resolved. A low resolution of the wake has, according to these results, a substantial part of pressure loss change. There are no clear signs of consequent mesh resolution dependency upstream the wake. The mesh resolution of the upstream manufacturing features does not seem to be essential. The changes at wake indicate that the importance of using the same number of nodes for all simulated domains, particularly at trailing edge, to allow comparison. Similar behaviour is obtained for k-ω SST.

The three turbulence models examined have similar characteristics for real and nominal sectors. Figure 26 shows a nominal regular sector with the turbulence models rk-ε k-ω SST and k-ω SST with transition. It is at the vane where the variations initiate.

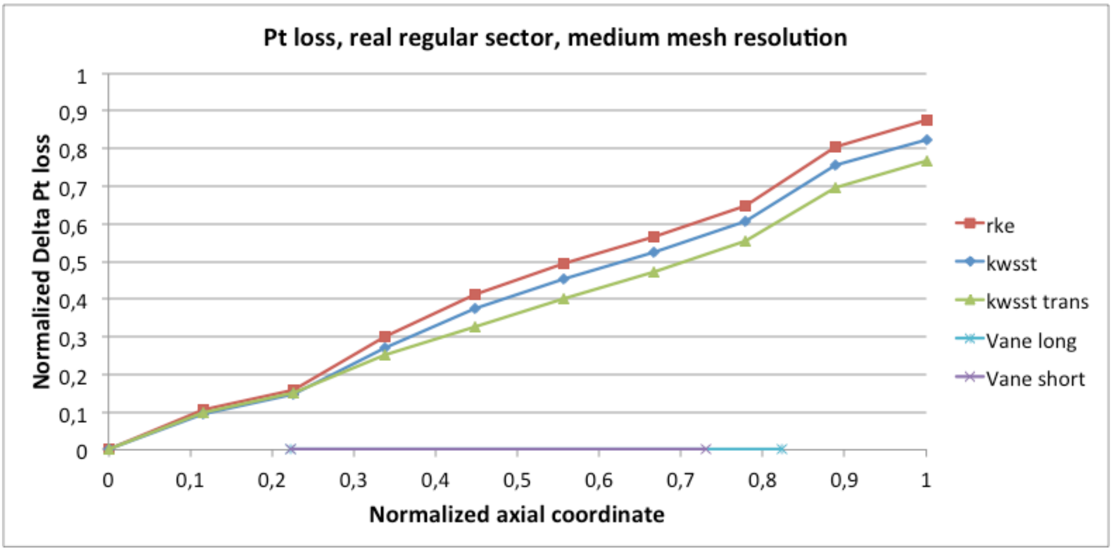


**Figure 26. Pressure loss for real sector with turbulence models rk-ε, k-ω SST and k-ω SST with transition.**

## 5.3 Analyses of Repaired Real Sectors

Analyses of sectors with repaired manufacturing features were made to explore the magnitude of the effect of various manufacturing features. A total pressure loss plot showing the delta between nominal and real sectors can generally be divided into three parts; loss increase upstream, along and downstream vane.

### 5.3.1 Manufacturing Features at Upstream Shroud

Figure 27 illustrates a cross-section of the modifications made, where the repaired model is relatively similar to nominal geometry. The left bump is the circular weld and the right one is the circular weld preparation.

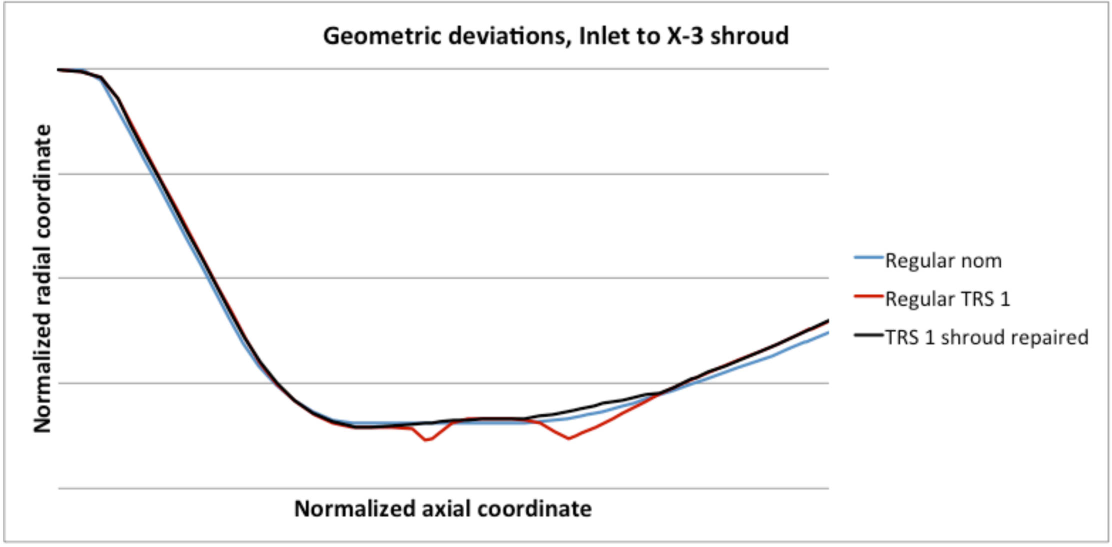


**Figure 27. Reparation of upstream shroud features.**

There is no low velocity in the actual area when these features are removed. It is a shape similar to a nominal model. The aerodynamic effects of circular welds and weld preparations at upstream shroud are noticeable in Figure 28. The figure shows where total pressure losses are developed compared to losses for a nominal geometry. The delta is explained by the difference in pressure loss for various geometries compared to the loss for nominal geometry. "Regular nom mfg" is a nominal geometry with weld preparations but no welds. Compare this to the real “Regular H” that has both welds and weld preparations. There is a steady increase of pressure loss because of these features up to an axial length of around 0.2, where the vane starts. “Regular nom mfg” does not have circular welds, hence does it not have this first loss increase. Eventually, “Shroud repaired” that has got none of these upstream shroud features do not have a loss increase at all. Except the no existing loss increase after inlet, “Regular H” and “Shroud repaired” have similar results after X-3.

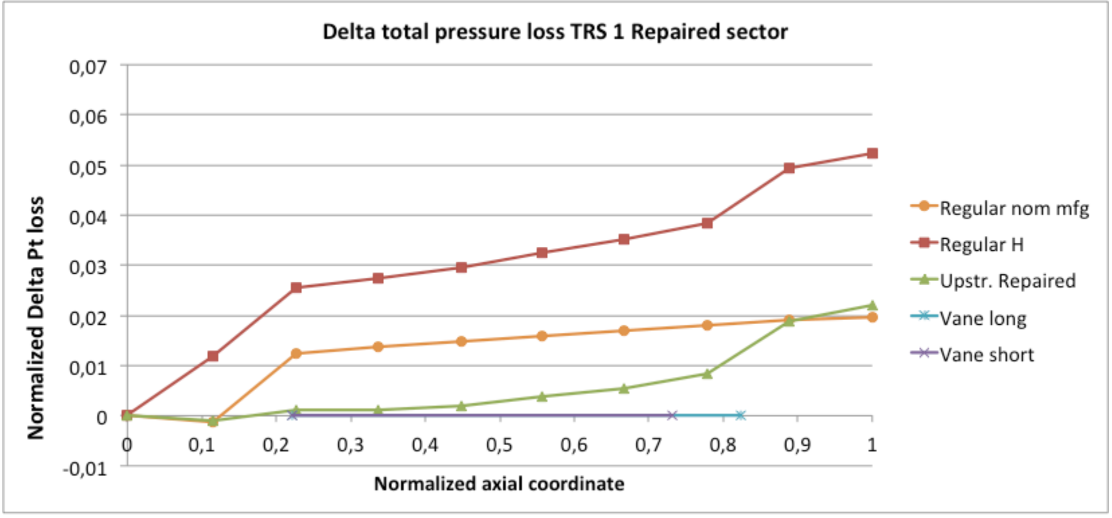


Figure 28. Pressure loss reduction when upstream manufacturing features were repaired.

**Potential loss (caused by manufacturing) reduction: 58%**

By repairing the weld and weld preparation at upstream shroud, the loss increase (from nominal) can be reduced.

There is a variation at vane span around 0.9 for nominal, real TRS and shroud repaired real TRS. The case with the repaired shroud features has almost as high total pressure as the nominal one. Consequently, it is obvious that total pressure losses can be related to upstream shroud features.

The only noticeable difference when looking at total pressure contour plots at TRS outlet, is the area of low-pressure at shroud. For nominal geometry it is fairly small, but it is increasing when upstream shroud features are present. The reparation of the real sector reduced this area clearly.

### 5.3.2 Manufacturing Features at Downstream Shroud

Figure 29 illustrates the geometric differences when the plate is repaired. This reparation for a deformed plate was a complex process, to not add new unwanted features. This is why the repaired shape is not following a nominal shape, the actual reparation turned out to be mostly a deletion of a circular weld.

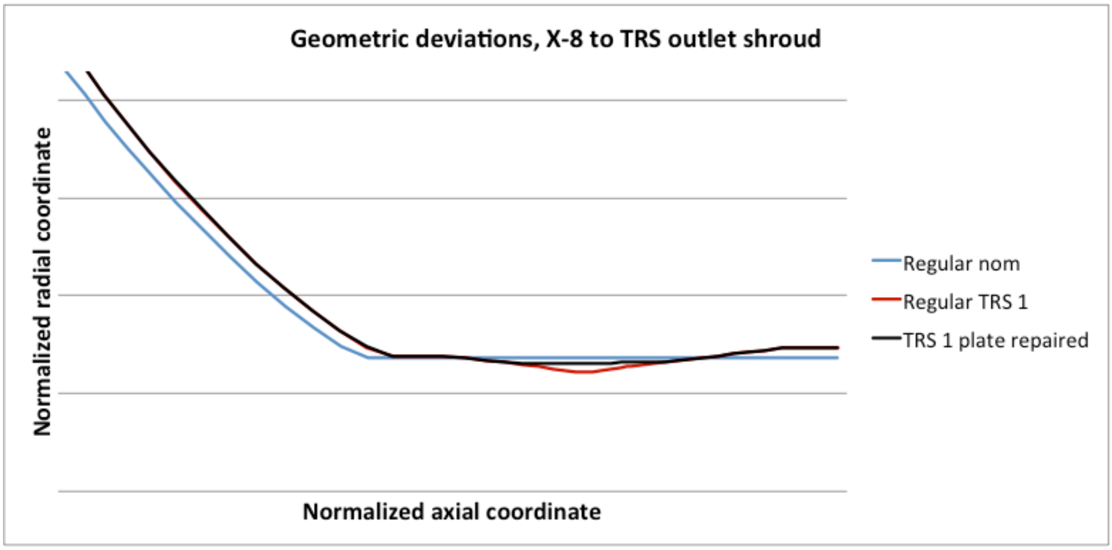


**Figure 29. Reparation of downstream shroud features at sector centre.**

It is obvious that the deviations are low at sector centre, but they are higher between the vanes. Figure 30 shows therefore the deviations between the vanes. It is still only the weld that is repaired, not the entire deviating surface.

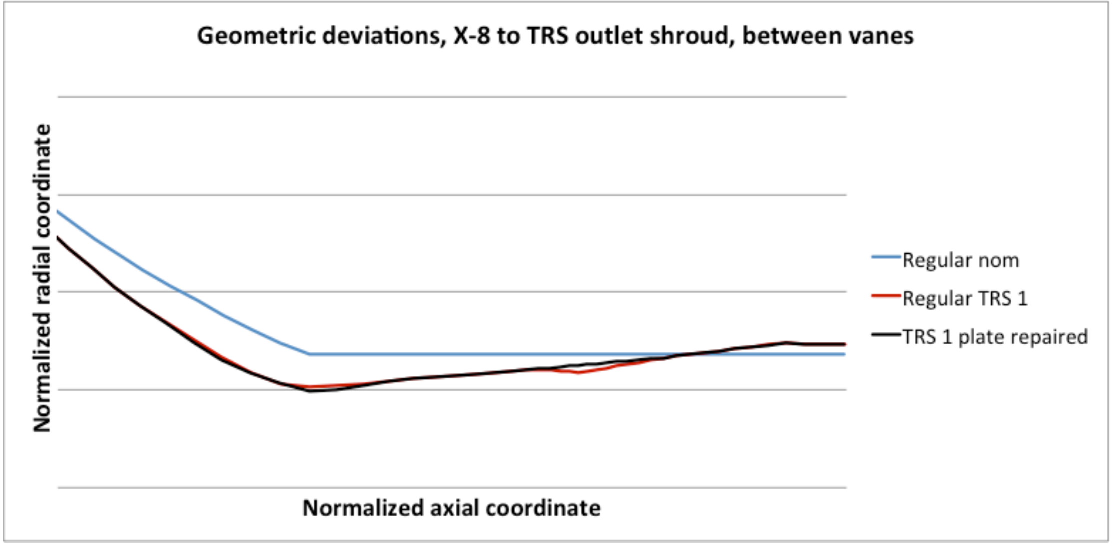


**Figure 30. Reparation of downstream shroud features, between vanes.**

In Figure 31 is the total pressure loss delta when the downstream shroud features are repaired. The improvement is not as big as it could be, but a small improvement is visible. If the shape would be similar to “Regular nom mfg”, there would probably be only a small loss increase after trailing edge.

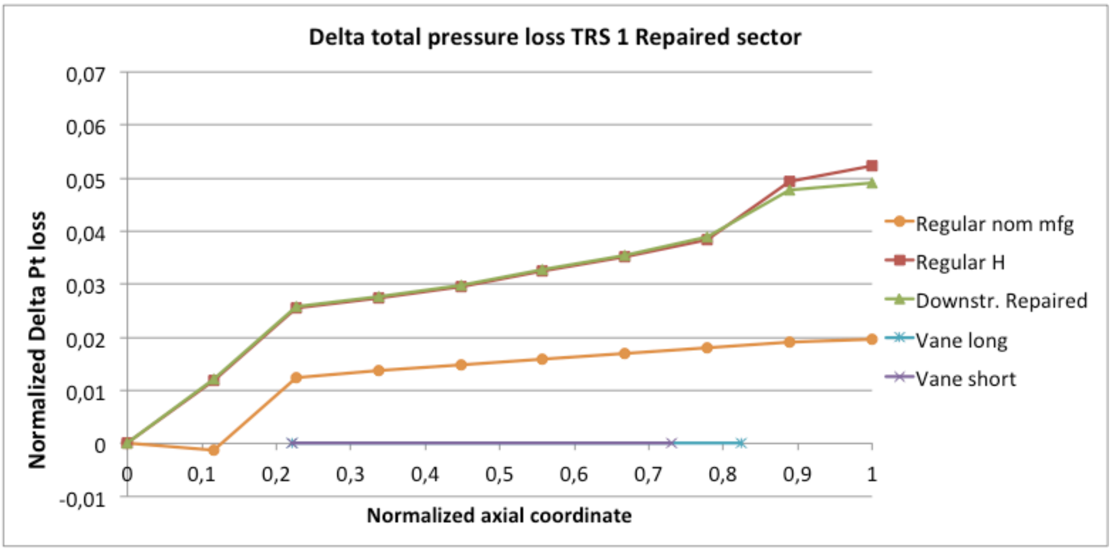


Figure 31. Pressure loss reduction when downstream manufacturing features were repaired.

**Potential loss (caused by manufacturing) reduction: 6%**

This downstream shroud reparation leads to a small reduction of delta. The belief of the plate reparation was to obtain results similar to "Regular nom mfg" downstream the vane. In other words, to receive a more horizontal loss graph close to outlet. In the cross-section Figure 29 and Figure 30 it is clear that the reparation was not as close to nominal surface as expected.

### 5.3.3 Manufacturing Features on Vane – Welds on Vane

The case where the welds on vane were repaired does not show any noticeable differences. In other words, with turbulence model rk-ε the welds on vane do not increase pressure loss. k-ω SST with transition was also simulated, but still no remarkable aerodynamic effects of welds on vane were found. Figure 32 shows the total pressure loss, where the graph of the repaired sector intersects with the graph of the original sector.

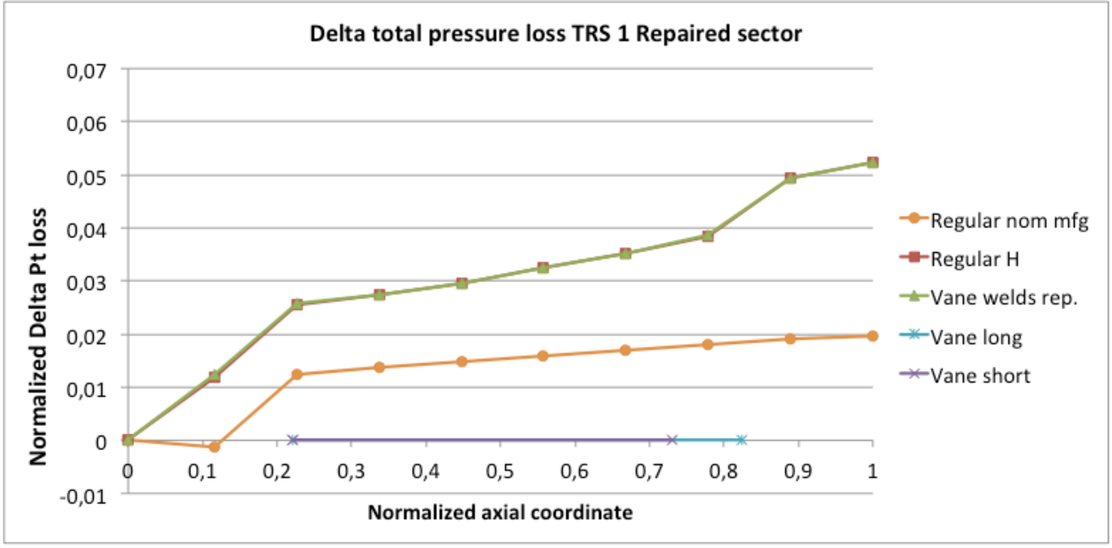


Figure 32. Pressure loss when welds on vane were repaired.

**Potential loss (caused by manufacturing) reduction: 0%**

The reparation of welds on vane leads to no delta loss reduction. For all generated data there was barely no difference for welds on vane and the repaired weld, therefore are no more results presented here.

## 5.4 Identification of Total Pressure Losses

Chapter 5.3 identified the effects of removing certain manufacturing features. It is a way of finding the effects of each feature. One origin of total pressure losses is manufacturing features, which is relatively local. One other origin could be the global deviations and displacements due to manufacturing. However, Figure 33 shows how much of the losses that could be reduced if there were no local manufacturing features. The remaining losses should therefore rise from other origins. By removing two features studied in previous chapter, the remaining loss is minor.

1. Remove weld and weld preparation at upstream shroud. Then it is possible to come down to the "Shroud repaired" value, which is close to nominal.
2. Along the vane there is a slight loss increase from nominal, see "Shroud repaired". For "Regular nom mfg" there is also an increase, which can be explained by the axial weld preparations on both sides of the vane at shroud. "Shroud repaired" has beyond weld preparations, also axial welds.
3. For real sectors there is generally a loss increase downstream the vane. "Regular nom mfg" has a fairly linear loss increase along the vane that continues also after the vane, so no particular losses appear in the wake. If real sectors would have no upstream features (1.) and a loss similar to "Regular nom mfg" in the wake, the loss due to manufacturing could be reduced by approximately 75%.

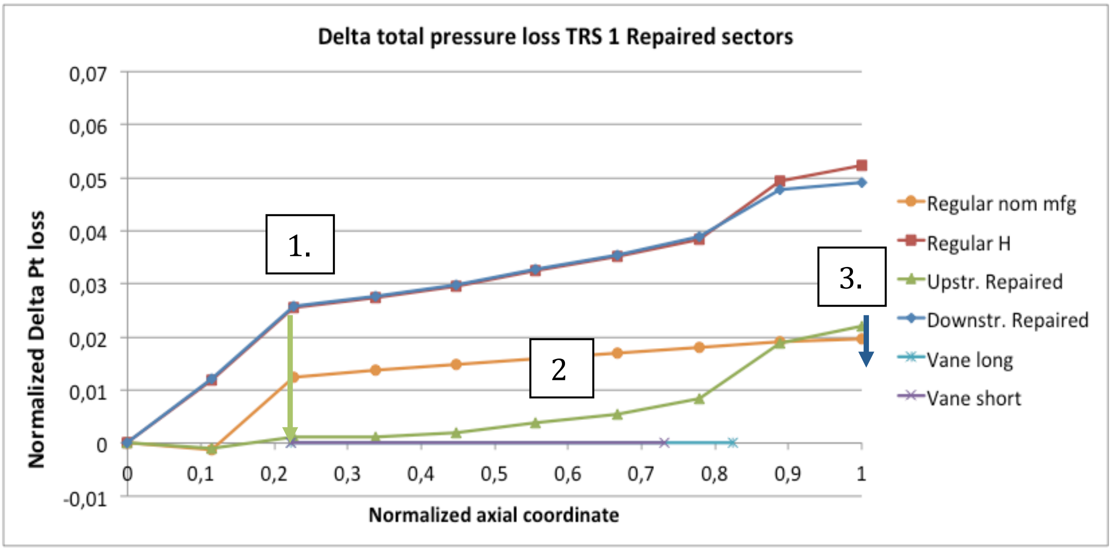


**Figure 33. 1.) Removing of upstream manufacturing features. 2.) Loss increase due to axial manufacturing features. 3.) Potential loss reduction without any unwanted geometric deviations at downstream shroud.**

Only a small loss remains, if steps 1-3 are accomplished. These remaining losses can be explained as the effect of the axial manufacturing features that are modelled in "Regular nom mfg". "Regular nom mfg" do not have any global deviations, so this means that global deviations or displacements do not have a noticeable effect on the results of the studied real sector. Though, there are variations among sectors as for the regular sectors in Figure 23 that could probably also be explained by other factors than only manufacturing features.

## 5.5 Design and Manufacturing Improvement

The discussions at the improvement meeting resulted in a table that covers problem, root causes, potential solutions and some evaluation of the suggested solutions for the manufacturing features. This chapter only presents a summary of the evaluation, the complete table can be found in the GAS document [10].

### 5.5.1 Root Causes for High Total Pressure Loss

The evaluated features that were identified to increase performance losses are assumed to be root causes to the pressure loss problem. The procedure of welding and weld preparations are different, hence these features are considered separately. The root causes are listed below.

- **Weld at upstream shroud**
  Feature indicated to generate low velocity boundary layer.
- **Weld preparation at upstream shroud**
  Feature indicated to generate low velocity boundary layer.
- **Welds on vane**
  Not any effects on losses were identified, but considered anyway since it is an unknown feature.

### 5.5.2 Potential Solutions of Root Causes

There are several approaches how to reduce the effects of the root causes. Potential solutions were discussed at the improvement meeting.

**Solutions for weld and weld preparation at upstream shroud:**

- **Turn with lathe machine**
  Use lathe machine with TRS rotating to level out the features.
- **Grind manually**
  Features are levelled out manually.
- **Move axial location of weld and weld preparation upstream**
  Move the axial location of the weld to a less affecting location, e.g. to the pocket next to the low-pressure turbine.
- **Reduce weld height when welding**
  Weld height depends on material thickness. Though, the height could be lower in the gas channel and higher on the side, which is not exposed by mass flow.
- **Change radius of weld preparation machining**
  A higher radius would result in a lower peak of the weld preparation (Figure 34).

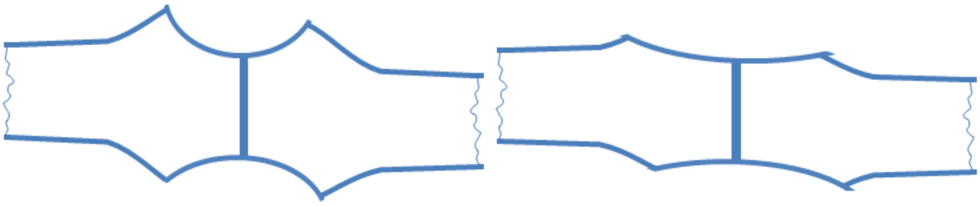


Figure 34. Small and big radius of weld preparation machining.

- **Reduce material thickness at weld prep preparation**
  When the manufacturing process is reliable, the exaggerated thickness and hence the height of the weld preparations could be decreased.

**Solutions for welds on vane:**

- **Grind manually**
  Same solution is possible as for upstream welds and weld preparations, but on vane.
- **Move location of weld**
  Move to a location where they have less influence, e.g. leading edge.
- **Reduce weld height when welding**
  Same solution is possible as for upstream shroud welds.

### 5.5.3 Evaluation of Solutions

In the scoring matrix [10] are benefit on aerodynamic performance, cost and risk of implementation considered. Also, each solution is assessed if it is suitable as a short-term or long-term improvement. Some solutions are short-term solutions, which are not optimal for serial production, and some are long-term, which need more comprehensive changes in design or manufacturing. The cost and risk criteria considered the manufacturing process and the requirements of the product.

**Weld at upstream shroud**

As a short-term solution, manual grinding is an alternative. For a more long-term solution it could be worth to consider a reduction of the weld height when welding. An overlap of materials would need a small redesign of the interfaces but could in long-term become beneficial. To move the location of the weld, an extensive redesign needs to be made.

**Weld preparation at upstream shroud**

Some results are valid for both the weld and the weld preparation. However, a higher radius of the weld preparation machining could be a simple way of lowering the height of the weld preparation. Also, when the manufacturing process is reliable enough, the thickness of the material before machining is certainly possible to reduce. This would also result in a lowered weld preparation height.

**Plate deformation at downstream shroud**

Thicker materials are not an option to consider unless there are no other alternatives. On the other hand, it could be possible to over-bend the material at a low cost and risk.

### Welds on vane

A low benefit on performance makes an improvement here not urgent at the moment. If other evaluations would show that welds on vane have a higher effect on aerodynamic performance than stated here, an improvement could be considered. Grinding is only possible to do manually, because the displacements do the use of a turning lathe difficult and could result in thicknesses under tolerances. A reduced weld height is hard to achieve, since it already is extremely low.

# 6 Discussion

The design practice method development, the CFD analyses and the potential improvements are discussed separately. For the design practice method, some advantages and disadvantages are described. The CFD analysis discussion considers the effect of manufacturing features and also the reliability of the results obtained. At the end, the potential improvements for a reduced pressure loss are discussed.

## 6.1 Design Practice Method

The results from this thesis can work as guidelines for design and manufacturing development of future TRSs. During product development, when certain manufacturing features need to be aerodynamically evaluated, the developed design practice method is useful. If the overall performance of the full manufactured TRS is wanted, the existing design practice method from Gustafsson [6] is more accurate. In that method, there are no fictive geometries at periodic surfaces and the interaction between all guide vanes is modelled. This gives a more reliable result for the TRS performance. The design practice method developed in this thesis is more accurate when sectors need to be compared under the same circumstances, without the effects of the surrounding sectors, or when a higher degree of detail is necessary.

Pros:

- Detailed analyses of real geometries are possible and the design practice method allows higher resolution of surface and volume meshes than full 360 degrees TRS analyses
- Obtained data from CFD is comparable to all individual sector analysis data done in the design phase
- Allows evaluation of the magnitude of manufacturing features for the aerodynamic performance
- Identical boundary surfaces for the CFD domain simplifies creation and analyse of the CFD domain
- Boundary conditions are equal for all cases, which make the sectors comparable

Cons:

- Real geometry with high geometric deviations creates unwanted features in buffer zones at periodic surfaces
- Buffer zones hide possible manufacturing features close to periodic surfaces
- Not optimal to determine the performance of a full TRS, since the sector wise simulations neglect the interaction of the guide vanes
- Relatively time consuming in terms of manual work

## 6.2 CFD Analyses

The results chapters described in detail the aerodynamic effects of the upstream shroud manufacturing features. These features stand for the most substantial part of the overall total pressure loss. The loss is more sensitive to features close to inlet, compared to features further downstream. One reason is that the boundary layer generally becomes thicker further down the gas channel. The downstream manufacturing features do not have as much impact on the total pressure loss, as if they were positioned upstream were the boundary layer is thinner.

Repairing real sectors has various complexities depending on type and location of a manufacturing feature. The upstream shroud features are fairly simple to repair, since the surrounding surface is flat. The downstream circular weld is also simple, but when a bigger area is deviating, it is more complex. The downstream shroud reparation did not represent a nominal surface, instead mostly a removal of the circular weld. Therefore, the results are not entirely reliable considering that the intention was to simulate how also the extensive deviation affected the results. However, the removal of the weld gave a small loss reduction.

A weld on a vane has such a small height, which makes it difficult to repair. This would also be true in reality of the weld is to be manually grinded. It is not certain that it would improve the performance, because the weld already has a small height. If the weld height would be higher, it could be more important to consider welds on vane.

The mesh resolution trend is not entirely reliable due to the inconsistency of the number of nodes on trailing edge. The results seem to be more dependent on the wake (trailing edge) resolution than anywhere else. Though, all other simulated cases have all used the same number of nodes on the trailing edge, hence this variation risk should be eliminated. Consequently, it is important to have as similar mesh as possible to obtain reliable results.

## 6.3 Design and Manufacturing Improvements

Aircraft engines do generally stay in service for many years, and a particular TRS version will be produced during a long time. This opens for long-term improvements to be implemented. The short-term solutions could become very costly if they are conducted during a long time. The need of improvements also has to be considered. If the aerodynamic requirements are already fulfilled and there are no signs of exceeding them, it would not be beneficial to spend money on improvements. On the other hand, an implementation of some of the suggested improvements could increase the requirement margins and the aerodynamic performance would become more robust. If the most effecting features are improved, other features do not need to be evaluated to be sure that requirements are fulfilled.

All possible root causes where not included in the evaluation, only those manufacturing features that were analysed with CFD.

# 7 Conclusions

The main conclusions for this study are listed below.

- The design practice method offers a method for detailed evaluation of aero performance for TRSs. Higher resolution on surface and volume mesh allows deviations from nominal geometry to be analysed in detail.
- Location and shape of manufacturing features decide the impact of a feature. Features perpendicular to the flow direction are more critical than the ones along the flow direction. Also, features close to the TRS inlet are in this study more critical than features back at TRS outlet.
- By repair of circular weld and weld preparation at upstream shroud, the aerodynamic loss due to manufacturing can be reduced by around 60%
- Welds on guide vane with a low weld height do not increase the total pressure loss noticeable according to cases analysed.
- Manufacturing features studied in this thesis can explain most of the aerodynamic losses due to manufacturing. Other geometric deviations are not as critical.
- Design and manufacturing improvements to reduce the influence of manufacturing features are possible, but need to be assessed if they are needed to fulfil requirements.

# 8 Future Work

A more comprehensive mesh dependency study could be conducted. It would be beneficial if the number of mesh nodes could be reduced without losing the influence of manufacturing features. The calculation time would be reduced. The approach of repairing manufacturing features could be used on more sectors and other features than studied in this thesis. Real surfaces can also be combined with nominal surfaces to create repairs more similar to nominal surfaces. Future work is also to put all individual sectors together and simulate it as a full 360 degrees TRS. The overall pressure loss would then be compared to the results of simulating all sectors individually. The effect that sectors have on other sectors would be determined. At GAS, this thesis provides useful information for future development projects.

# 9 References

1. GKN Aerospace, 2013, [cited 2013 December 1], Available from: http://www.gkn.com/aerospace/

2. NASA, *How does a jet engine work?*, 2010, [cited 2013 December 1], Available from: http://www.grc.nasa.gov/WWW/k-12/UEET/StudentSite/engines.html

3. White, F.M., *Fluid Mechanics*, Sixth Edition, McGraw-Hill, 2009

4. ANSYS, *ANSYS FLUENT 12.0 Theory Guide*, Chapter 4, ANSYS Inc., 2009

5. Andersson, B., Andersson, R., Håkansson, L., Mortensen, M., Sudiyo, R., van Wachem, B., *Computational Fluid Dynamics for Engineers*, Ninth Edition, Chalmers University of Technology, 2013

6. Gustafsson, B., Design Practice Method, GKN Aerospace Sweden AB, VOLS:10179741, 2013 (Internal document)

7. Ström, L., Design Report, GKN Aerospace Sweden AB, VOLS:10142969, 2013 (Internal document)

8. Linde, O., Ström, L., Design Practice Method, GKN Aerospace Sweden AB, VOLS:10190212, 2013 (Internal document)

9. Linde, O., Ström, L., GKN Aerospace Sweden AB, VOLS: VOLS:10183605-000,001,005,-01, 2013 (Internal documents)

Final presentation at GAS and this report can be found under: VOLS:10190214

**Figures:**
The author created most figures in this report. For all other figures, sources are presented in the caption.