Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

Lise-Marie Dahlby
2016

Master of Science in Engineering Technology
Space Engineering

Luleå University of Technology
Department of Engineering Sciences and Mathematics
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

Master’s Thesis Report

Author: Lise-Marie Dahlby
Supervisors: Prof. Lars-Göran Westerberg, Luleå University of Technology
Linda Ström, PhD, GKN Aerospace Engine Systems, Sweden
Pär Nylander, MSc, GKN Aerospace Engine Systems, Sweden

5/11/2016
Abstract

In this Master thesis, the prediction of the aerodynamic performance on a low pressure turbine outlet guide vane is investigated. The standard approach at many aerospace companies to predict aerodynamic performance of gas turbine components like the guide vane is to use fully turbulent models in CFD analyses. However, recent tests show that the boundary layer on the guide vane often is initially laminar before transitioning into turbulent. Transition may occur through a process with natural instability, through bypass processes or through laminar separation. The transition mechanism depends, among other things, on the Reynolds number of the flow, the pressure gradient and the freestream turbulence intensity level.

The main focus of this Master thesis has been to investigate the aerodynamic performance parameters like flow separation and pressure loss (both 2D and 3D loss) by applying three different transition models in CFD. The CFD predictions were further compared to test data from the test rig at Chalmers University of Technology. In order to study the impact of the mesh resolution on the results, two different meshes were also used.

It was found that the transition models studied in this Master thesis show good agreement with test data in terms of vane static pressure loadings, wake pressure profiles, 2D pressure losses and also predicts laminar to turbulent transition by a laminar separation, like the test data shows. It was also found that the differences in mesh resolution studied here do not affect the results much, in terms of pressure loss predictions. A low-resolution mesh might need refinement if there are convergence issues, however.
Acknowledgements

I would like to express my gratitude to my supervisors Linda Ström and Pär Nylander at GKN Aerospace Sweden AB for all their help and support throughout this thesis work. I also want to thank my supervisor, Professor Lars-Göran Westerberg at Luleå University of Technology. I wish to thank Markus Nordahl, Manager Engineering at dept. 9642 for giving me the opportunity to do my master’s thesis at GKN. I also want to thank Euodia Krüger for taking the time to help me with CFX and post-processing scripts. Additionally, I would like to thank all engineers at dept. 9642 for their warm welcome and for all their help and advice during this thesis work.

Last, but certainly not least, I would like to thank my friends who have been there for me throughout this whole journey and especially my family – Thomas, Maria, Christoffer, Martina – for their constant love and support and their unwavering belief that I can do anything I set my mind to.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

MAY 11, 2016

Nomenclature

<table>
<thead>
<tr>
<th>Term</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S$</td>
<td>Absolute value of strain rate</td>
</tr>
<tr>
<td>$Cx$</td>
<td>Axial chord length [m]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density [kg/m$^3$]</td>
</tr>
<tr>
<td>$q_{in}$</td>
<td>Dynamic pressure [Pa]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity [Pa·s]</td>
</tr>
<tr>
<td>$U$</td>
<td>Freestream mean velocity [m/s]</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity [m$^2$/s]</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Intermittency</td>
</tr>
<tr>
<td>$\lambda_\theta$</td>
<td>Pressure gradient parameter</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Turbulence dissipation rate [J/(kg·s)]</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulence kinetic energy [J/kg]</td>
</tr>
<tr>
<td>$T_U$</td>
<td>Freestream turbulence intensity [%]</td>
</tr>
<tr>
<td>$Re_{\theta t}$</td>
<td>Transition onset momentum-thickness Reynolds number based on freestream conditions</td>
</tr>
<tr>
<td>$\tilde{Re}_{\theta t}$</td>
<td>Local transition onset momentum-thickness Reynolds number obtained from transport eq.</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Specific dissipation rate [1/s]</td>
</tr>
<tr>
<td>$P_s$</td>
<td>Static pressure [Pa]</td>
</tr>
<tr>
<td>$Re_\nu$</td>
<td>Strain rate/vorticity Reynolds number</td>
</tr>
<tr>
<td>$P_t$</td>
<td>Total pressure [Pa]</td>
</tr>
<tr>
<td>$y$</td>
<td>Wall distance [m]</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>Wall shear stress [Pa]</td>
</tr>
</tbody>
</table>

Abbreviations

<table>
<thead>
<tr>
<th>Term</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADP</td>
<td>Aero Design Point</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CTH</td>
<td>Chalmers University of Technology</td>
</tr>
<tr>
<td>ER</td>
<td>Wall-Normal Expansion Ratio</td>
</tr>
<tr>
<td>GAS</td>
<td>GKN Aerospace Sweden</td>
</tr>
<tr>
<td>LCTM</td>
<td>Local Correlation based Transition Modelling</td>
</tr>
<tr>
<td>LPT</td>
<td>Low Pressure Turbine</td>
</tr>
<tr>
<td>OGV</td>
<td>Outlet Guide Vane</td>
</tr>
<tr>
<td>TEC</td>
<td>Turbine Exhaust Case</td>
</tr>
<tr>
<td>TRF</td>
<td>Turbine Rear Frame</td>
</tr>
</tbody>
</table>
Confidentiality

This is an open version of a confidential report and therefore, due to being GKN Aerospace Sweden AB Proprietary information, the test data will not be shown. Some axes and scales in plots and figures have also been either removed or normalized.
# Table of contents

Abstract ............................................................................................................................... i
Acknowledgements ........................................................................................................... ii
Nomenclature .................................................................................................................... iii
Abbreviations ................................................................................................................... iii
Confidentiality ................................................................................................................... iv
1 Introduction ................................................................................................................... 2
   1.1 Background .............................................................................................................. 2
   1.2 Purpose ................................................................................................................... 2
   1.3 Research Questions ............................................................................................... 3
   1.4 Limitations ............................................................................................................. 3
   1.5 CTH-Rig ................................................................................................................. 3
2 Theory ........................................................................................................................... 5
   2.1 Guide Vane Aerodynamics ..................................................................................... 5
   2.2 Boundary Layers .................................................................................................. 5
   2.3 Transition .............................................................................................................. 6
   2.4 Transition Mechanisms ........................................................................................ 6
      2.4.1 Natural Transition ............................................................................................ 6
      2.4.2 Bypass Transition ........................................................................................... 6
      2.4.3 Separation Induced Transition ....................................................................... 6
   2.5 Computational Fluid Dynamics (CFD) .................................................................. 7
   2.6 Turbulence Modelling ............................................................................................ 7
   2.7 Transition Modelling .............................................................................................. 7
      2.7.1 ANSYS Fluent ................................................................................................ 8
      2.7.2 ANSYS CFX .................................................................................................. 8
3 Method .......................................................................................................................... 10
   3.1 Test Matrix, Test Data and Simulation Overview .................................................... 10
      3.1.1 Test Data ......................................................................................................... 11
   3.2 CFD Analysis ......................................................................................................... 11
      3.2.1 Geometry ........................................................................................................ 11
      3.2.2 Boundary Conditions ...................................................................................... 12
      3.2.3 Analysis Settings ............................................................................................ 12
   3.3 Mesh Specifications ............................................................................................... 13
   3.4 Evaluation Parameters .......................................................................................... 15
      3.4.1 Flow Quality – Separation and Streamlines ..................................................... 15
      3.4.2 Vane Characteristics - Static Pressure Loadings and X-wall Shear Stress ....... 15
      3.4.3 Total Pressure Loss - 2D and 3D Wake Profiles .............................................. 16
4 Results ........................................................................................................................... 17
   4.1 Re 2.74·10^5 ............................................................................................................ 17
      4.1.1 Overview ......................................................................................................... 17
      4.1.2 10 Degrees ..................................................................................................... 20
      4.1.3 20 Degrees (ADP) ......................................................................................... 22
1 Introduction

This introductory chapter intends to present the background and the purpose of the thesis, as well as state the research questions that were in focus during the project.

1.1 Background

This master thesis is a study of aerodynamic performance predictions on a low pressure turbine (LPT) outlet guide vane (OGV), using different turbulence schemes to model transition from laminar to turbulent flow. The guide vanes are positioned in a structure called Turbine Rear Structure (TRS), Turbine Exhaust Case (TEC) or Turbine Rear Frame (TRF), which is placed downstream of the LPT. The TRS/TEC/TRF is the structure on which the engine is mounted to the aircraft wing. The notation TEC will be used henceforth in the thesis. Aerodynamically, the OGV serves to reduce the swirl of the outlet flow from the LPT [1]. Figure 1 shows an illustration of a jet engine with the TEC circled in red.

![Figure 1: Illustration of a jet engine with the TEC circled in red. Source: GKN Aerospace (www.gknaerospace.com)](image)

1.2 Purpose

This study has been done because accurate and validated methods for predicting aerodynamic performance parameters, like pressure loss, separation and swirl, are needed. The standard approach to predict aerodynamic performance of OGVs at many aerospace companies is to use fully turbulent models in CFD (Computational Fluid Dynamics) analyses. However, recent tests suggest that the boundary layer on the vane surface is not fully turbulent, but laminar at first before transitioning into a turbulent flow.

Since the fully turbulent models cannot capture the transition, the prediction of the aerodynamic performance might not be completely accurate; especially not the physics of the flow characteristics. A transition model would be able to better capture the correct physics of the boundary layer growth, unlike a fully turbulent model. This is important when it comes to predicting aerodynamic performance like separation; a laminar and a turbulent boundary layer has very different separation behaviour, e.g. a laminar boundary layer is more sensitive to instabilities and separates more easily [2, 3]. To perform CFD analyses with a transition model would therefore be closer to the truth and would give a better prediction of the aerodynamic performance than a fully turbulent model.
In order to investigate the aerodynamic performance predictions, some available transition models in ANSYS Fluent and CFX have been used and compared to test data from Chalmers University of Technology (CTH).

1.3 Research Questions

The following questions will be answered during this project:

- Is the flow transitional in the test rig?
- Which transition model/CFD tool works best, in general?
- How accurate is CFD in comparison to test data?
- How does transition vary with flow angle and/or Reynolds number?
- How does the mesh resolution affect the results?
- How does the turbulence intensity affect transition?
- What should a best practise be for transition modelling for this application?

1.4 Limitations

Due to the time limit of this project and the many simulations that needed to be run, some limitations had to be set. One of these limitations was to use only three of the available transition models in ANSYS Fluent and CFX.

There are many factors that can affect how and at what point on a surface that transition from laminar to turbulent flow occurs; Reynolds number, inlet flow angle, freestream turbulence intensity and surface roughness to name a few. However, not all of these factors were studied. The Reynolds number and inlet flow angle were varied and only one case with different turbulence intensity was studied. Surface roughness was not considered in this project.

Simplified boundary conditions were used since post-test CFD boundary conditions were not available at the start of the thesis.

1.5 CTH-Rig

The CTH test rig is a large scale, low speed linear cascade with a channel height of 0.24 m. Four vanes are inserted in the channel in the current test. The test object, i.e. the vane, is a 2D version of a 3D OGV. A picture of the test rig is shown in Figure 2 along with a schematic picture of the test section [4].

![Figure 2: Linear cascade at CTH (left) and schematic of test section (right). Source: ref [4].](image)
The airflow in the rig can be adjusted to different velocities and the test section can be adjusted to different inlet angles. The airflow velocity was varied in the experiments, corresponding to flow conditions approximate to Reynolds numbers of $2.20 \cdot 10^5$ to $4.80 \cdot 10^5$ based on the freestream mean velocity ($U$) and axial chord length ($Cx$). For this characteristic length scale and velocity the Reynolds number reads:

$$Re = \frac{U \cdot Cx}{\nu}$$  \hspace{1cm} (1)

where $\nu$ is the kinematic viscosity. The inlet flow angles ranged from 10° to 30°. See Table 3 in section 3.2 for more details.
2 Theory

This chapter will present the reader with some theory regarding the aerodynamics of a guide vane and some theoretical background of transition. It will also give a short description of CFD as well as present some theory about transition modelling using CFD.

2.1 Guide Vane Aerodynamics

The guide vane is comparable to any other airfoil, with the same nomenclature used to describe its geometry. Figure 3 shows an airfoil representative of a guide vane, with all the names necessary for this thesis specified. The front part of an airfoil is called the leading edge, while the rear part is called the trailing edge. The flow direction in the figure is left to right and in order to describe the location in a fluid flow, “upstream” and “downstream” is often used. Upstream is in the direction of the leading edge and downstream is in the direction of the trailing edge. On the suction side of the vane, the flow velocity is high and the static pressure is low (negative), hence the name “suction side”. On the pressure side of the vane, the flow velocity is low and the static pressure is high. The wake is the region of low velocity and low total pressure behind the vane. The length of the vane is the distance between the leading and trailing edge and is called the chord [5].

![Figure 3: Illustration of an airfoil representative of a guide vane, with all names necessary for this thesis specified. Flow direction is left to right.](image)

2.2 Boundary Layers

Anderson [5] describes the boundary layer as "the thin region of flow adjacent to a surface, where the flow is retarded by the influence of friction between a solid surface and the fluid". Immediately closest to the surface of the body, the velocity is equal to zero relative to the surface due to air molecules that stick to the surface because of friction between the fluid and the surface. This is called the no-slip condition. The velocity then increases further away from the surface until it reaches 99% of the freestream velocity; this is defined as the edge of the boundary layer. The increase in velocity takes place over a very short distance since the boundary layer is so thin, thus the velocity gradients inside the boundary layer are very large. Often mentioned in boundary layer theory is the velocity profile; it is the variation of velocity inside the boundary layer and the slope of the velocity profile at the surface (or wall) governs the wall shear stress (i.e. the friction). The wall shear stress will be further considered in Chapter 3.4 [5].

A boundary layer can be laminar, transitional or turbulent. Laminar flow is smooth and layered while a turbulent flow is random and chaotic. Transitional flow occurs when a laminar flow starts to become unstable and transitions into a turbulent flow. Transition will be further discussed in Chapter 2.3. The differences between a laminar and turbulent boundary layer are significant and have a major impact on aerodynamics. For example, the wall shear stress is considerably higher
for a turbulent flow compared to a laminar flow which in turn means that the drag, and thus the pressure loss, is higher for a turbulent flow. A turbulent boundary layer is also thicker than a laminar one and is less inclined to separate from the surface of a body. Flow separation occurs when the flow over a surface produces an adverse pressure gradient (i.e. the pressure increases in the flow direction); the velocity closest to the wall is then reduced further until it reverses its direction and starts moving upstream. Flow separation is not desirable and can cause large pressure losses over an aerodynamic body [5].

2.3 Transition

Transition refers to the process of transformation from laminar to turbulent boundary layer flow. This can occur in several different ways: through natural, bypass, separation induced, wake induced or through cross-flow. These mechanisms depend on the freestream turbulence intensity, the pressure gradient, the geometry of the body in question, surface roughness and the Reynolds number to name a few [6, 7]. The main focus for this thesis work is on bypass and separation-induced transition however, since those are the mechanisms most likely to be present in the application in question. A closer description of natural, bypass and separation induced transition is presented in the following sections.

The effects of transition are many; it will, among other things, affect the aerodynamic performance, pressure loss and heat transfer of the body in question. In order to improve the efficiency and durability of a product, it is therefore important to be able to accurately predict the point and extent of transition [7]. As mentioned earlier, the separation behaviour between laminar and turbulent flow is also very different; e.g. a laminar boundary layer is more sensitive to instabilities and separates more easily, so it is important to capture the correct physics of the boundary layer growth [3].

2.4 Transition Mechanisms

2.4.1 Natural Transition

Natural transition occurs when the freestream turbulence intensity level is low, approximately <1% and the flow starts to become linearly unstable by way of Tollmien-Schlichting waves [8]. The growth of these waves is very slow and the change to fully turbulent flow can occur far downstream from the transition point. Since the turbulence intensity must be low for natural transition to occur, this mechanism is unlikely to be present for the application of LPT OGVs.

2.4.2 Bypass Transition

When the freestream turbulence level is high, >1%, bypass transition is likely to occur [5]. As the name suggests, this transition mechanism bypasses the first stages of transition that is present for natural transition (linear growth of two dimensional disturbances, i.e. Tollmien-Schlichting waves) and turbulent spots are developed directly in the boundary layer due to the freestream disturbances. When the freestream turbulence is high, the boundary layer is forced into transition further upstream than would be the case for natural transition [8].

2.4.3 Separation Induced Transition

A laminar boundary layer can separate due to an adverse pressure gradient, i.e. a pressure gradient that increases in the direction of the flow. Disturbances can then start to grow in the separated shear layer and trigger a transition to a turbulent boundary layer. Due to the enhanced mixing in the turbulent flow, the shear layer can then reattach to form a laminar separation bubble on the surface of the body in question. Laminar separation bubbles are, for example, typically
formed near the leading edge of thin airfoils and on gas turbine blades and the formation of such a bubble might affect the aerodynamic performance of these applications [8, 9].

The laminar separation bubble is a function of the Reynolds number and turbulence intensity; the separation bubble grows with decreasing Reynolds number or decreasing turbulence intensity [10].

2.5 Computational Fluid Dynamics (CFD)

The basic governing equations that describe the characteristics of aerodynamic flows are continuity, momentum and energy and they are either in integral or partial differential form. CFD is an advanced method for solving these equations numerically by simulating the fluid dynamics. This is done by first dividing the simulation domain (i.e. the volume which encloses the body of interest) into small control volumes (elements) and grid points (nodes) which create a mesh. Boundary conditions must then be specified in order to define the fluid properties at the boundaries of the simulation domain [5, 11].

The governing equations are then solved iteratively until convergence is reached and a solution is obtained. Convergence is measured by the residuals and the residuals are essentially the error, or imbalance, left after each solver iteration of the governing equations [12].

2.6 Turbulence Modelling

A turbulent flow is a three-dimensional unsteady phenomenon which contains many different turbulent length and time scales that all interact with each other. Most CFD calculations of turbulence use the so-called Reynolds-averaged Navier-Stokes (RANS) equations to simulate turbulent flow, which means that the turbulence is averaged to remove the need of simulating all scales of the turbulence spectrum [13]. A turbulence model is then needed to model the flow by using, for example, transport equations for the turbulence kinetic energy ($k$) and the turbulence dissipation rate ($\varepsilon$).

2.7 Transition Modelling

ANSYS Fluent and ANSYS CFX are two different CFD tools within the ANSYS software that both have broad modelling capabilities to model turbulence, heat transfer, etc. CFX is a vertex-based finite volume solver (assembles control volumes around each node in the mesh), while Fluent is a cell-based solver. In Fluent there is also the possibility to choose a coupled or segregated solver, while in CFX the solver is coupled [12].

There are several transition models available in both Fluent and CFX. The most widely used transition model in the industry at the moment is the Transition SST model, or the $\gamma$-$Re_\theta$ (Gamma-Theta) model. In recent years a new model has also been developed, called the Gamma model. These transition models are all part of a concept called Local Correlation based Transition Modelling (LCTM) [10].

Transition modelling is based on experimental correlations and a transition model needs to be coupled with a turbulence model (like the Shear Stress Transport model) that will be activated when the transition onset criteria is met. In order to understand transition modelling and the different models available in Fluent and CFX, some expressions need to be explained; the intermittency factor ($\gamma$) and the transition onset momentum-thickness Reynolds number ($Re_{\theta t}$). These two are key variables when it comes to transition modelling. The intermittency can be defined as “the fraction of time when the flow is turbulent” [7] and is equal to 1 in a turbulent flow and 0 in a laminar flow.
The transition onset momentum-thickness Reynolds number can be described as the point where the velocity profile first starts to deviate from an entirely laminar one [8]. It is a function of the turbulence intensity \( (Tu) \) and pressure gradient \( (\lambda_0) \) such that:

\[
Re_{Bl} = f(Tu, \lambda_0).
\]  

(2)

Another central variable in the transition models used in this thesis is the vorticity (or strain-rate) Reynolds number, \( Re_\nu \) [6]

\[
Re_\nu = \frac{\rho y^2}{\mu} \frac{\partial u}{\partial y} = \frac{\rho y^2}{\mu} S
\]  

(3)

where \( \rho \) is the density, \( \mu \) is the dynamic viscosity, \( y \) is the wall distance and \( S \) is the absolute value of the strain rate. \( Re_\nu \) is then used to link the experimental correlations in \( Re_{Bt} \) to the intermittency to trigger transition [6, 8].

2.7.1 ANSYS Fluent

In ANSYS Fluent 16.0 there are three different transition models available, the k-kl-\( \omega \) transition model, Intermittency transition model and the Transition SST model. For this study however, only the latter was used. The realizable k-\( \varepsilon \) model (rk-\( \varepsilon \)) is a fully turbulent one and was used for some cases in this thesis. Some theory of these models is briefly described in the following sections.

2.7.1.1 Realizable k-\( \varepsilon \) Model

The rk-\( \varepsilon \) model is a fully turbulent model, developed from the standard fully turbulent k-\( \varepsilon \) model. The rk-\( \varepsilon \) model has an alternative formulation of the turbulent viscosity along with a modified transport equation for the dissipation rate, \( \varepsilon \). “Realizable” means that the model satisfies some mathematical constraints on the Reynolds stresses, consistent with physics of a turbulent flow [7].

2.7.1.2 Transition SST Model

The Fluent Transition SST model is based on two transport equations, one for the intermittency and one for \( Re_{Bt} \), coupled with transport equations for \( k \) and \( \omega \) (the specific dissipation rate). Since \( Re_{Bt} \) is calculated outside the boundary layer using the freestream turbulence intensity and the pressure gradient (a non-local empirical correlation), the transport equation for \( Re_{Bt} \) is needed to transport this information into the boundary layer. Basically, the transport equation takes this non-local empirical correlation and transforms it into a local variable, \( \tilde{Re}_{Bt} \). This local variable can then be compared to \( Re_\nu \) to determine where in the flow the transition criterion has been met. At every point in the flow where \( Re_\nu > \tilde{Re}_{Bt} \), a source term in the intermittency transport equation is activated which then produces turbulence [8, 10].

The Fluent Transition model is coupled with the SST turbulence model (Shear Stress Transport), which is a combination of the k-\( \varepsilon \) and k-\( \omega \) models. It uses the k-\( \varepsilon \) model in the free flow and the k-\( \omega \) model close to the wall, in the boundary layer flow.

The complete theory of the transition model, with transport equations and model constants, is available in the ANSYS Help Guide [12].

2.7.2 ANSYS CFX

ANSYS CFX 16.0 has a transition model called CFX Transition SST or simply Gamma-Theta, as well as two reduced variants of this – the Gamma model and the Specified Intermittency model. The Gamma model is the latest one, developed by Menter et al. [10], and was therefore used along with the Gamma-Theta model.
**Transition SST Model (Gamma-Theta)**

The Gamma-Theta model in CFX is the same as the Transition SST model in Fluent, which is described in section 2.7.1.2.

### 2.7.2.1 Gamma Transition Model

The Gamma model is a simplified version of the Gamma-Theta model, with only one transport equation for the intermittency. This model has been made Galilean invariant, which the previous model was not. This means that the Gamma model can be applied to cases where the walls are moving relative to the coordinate system where the flow is computed [10].

The main difference from the Gamma-Theta model is the arguments $T_u L$ and $\lambda \theta L$ which goes into the correlation for $Re_{\theta t}$, so it now reads:

$$Re_{\theta t} = f(T_u L, \lambda \theta L).$$

Essentially, in the Gamma model, these arguments are approximated locally inside the boundary layer instead of outside the boundary layer as in the previous model. This makes the transport equation for the transition onset momentum thickness Reynolds number, $Re_{\theta t}$, unnecessary. Since this result in one less transport equation to solve, the Gamma model supposedly takes less computation time than the Gamma-Theta model [8, 10].
3 Method

This chapter will describe the method used during the thesis work. It will present the test matrix and go through the geometry and boundary conditions used for the simulations. This chapter will also describe some of the analysis settings for the CFD models and present the different meshes.

3.1 Test Matrix, Test Data and Simulation Overview

Table 1 shows a test matrix for this study, where the bold text indicates the most relevant cases and the cells in blue indicate for which cases there are test data available. Reynolds number $2.74 \times 10^5$ and inlet flow angle 20 degrees are conditions representative of the aerodynamic design point (ADP) and was therefore the most important case to study. The ADP represents the conditions for which the application is designed for its highest efficiency. For Reynolds number $2.74 \times 10^5$, all angles were simulated and all transition models were used with both meshes. For the other Reynolds numbers, not all angles were run, nor all models or meshes.

Table 2 is an overview of the simulations performed during this thesis. It shows which mesh, CFD tool (ANSYS solver) and turbulence model was used for each Reynolds number and flow angle. If a mesh and turbulence model was used for a particular angle it is indicated by H (high-resolution mesh) or L (low-resolution mesh), otherwise it is left blank. If convergence was questionable for any simulation, the cell is shown in red.

<table>
<thead>
<tr>
<th>Angle/Re</th>
<th>$2.20 \times 10^5$</th>
<th>$2.74 \times 10^5$</th>
<th>$3.20 \times 10^5$</th>
<th>$4.80 \times 10^5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>CFD</td>
<td>CFD, Test data</td>
<td>CFD</td>
<td>CFD</td>
</tr>
<tr>
<td>20</td>
<td>CFD, Test data</td>
<td>CFD, Test data</td>
<td>CFD, Test data</td>
<td>CFD, Test data</td>
</tr>
<tr>
<td>25</td>
<td>CFD, Test data</td>
<td>CFD, Test data</td>
<td>CFD, Test data</td>
<td>CFD, Test data</td>
</tr>
<tr>
<td>30</td>
<td>CFD</td>
<td>CFD, Test data</td>
<td>CFD</td>
<td>CFD</td>
</tr>
</tbody>
</table>
Table 2: Overview of performed simulations in terms of Reynolds number, angle, mesh, CFD tool and turbulence model. “H” indicates the high-resolution mesh and “L” indicates the low-resolution mesh. The cells marked in red are simulations where the convergence is questionable.

<table>
<thead>
<tr>
<th>Re</th>
<th>Angle</th>
<th>Fluent</th>
<th>CFX γ-θ</th>
<th>CFX γ</th>
<th>rk-ε</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.20×10⁵</td>
<td>10</td>
<td>H</td>
<td>H</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>H, L</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td>2.74×10⁵</td>
<td>10</td>
<td>H, L</td>
<td>H, L</td>
<td>H, L</td>
<td>H</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>H, L</td>
<td>H, L</td>
<td>H, L</td>
<td>H</td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>H, L</td>
<td>H, L</td>
<td>H, L</td>
<td>H</td>
</tr>
<tr>
<td>3.20×10⁵</td>
<td>10</td>
<td>H</td>
<td>H</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>H, L</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>H, L</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td>4.80×10⁵</td>
<td>10</td>
<td>H</td>
<td>H</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>H, L</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td></td>
</tr>
</tbody>
</table>

3.1.1 Test Data

The data was preliminary at the time of this project, since it was still being processed and analysed. The data available was oil flow visualization, pressure and velocity at inlet and outlet (measured by multi-hole pressure probes) and static pressure on the vane. The oil flow visualization is paint applied to the vane and was used to analyse the flow quality on the vane surface.

3.2 CFD Analysis

3.2.1 Geometry

Figure 4 shows the simulation domain along with the guide vane, with the proper names of all surfaces displayed. The translational periodic boundaries means that flow exiting one boundary will enter the other. The inlet is set as velocity inlet and the outlet set as pressure outlet.

The length of the domain is in the x-direction and an outlet evaluation plane named x-out is located some distance downstream of the vane. The inlet evaluation plane is located upstream of the vane. This is in accordance with the experimental setup in the CTH-rig. The domain has a height (y) and a width (z), with z = 0 set at mid span of the vane.
3.2.2 Boundary Conditions

The inlet boundary conditions are tabulated in Table 3 along with freestream velocity and respective Reynolds number. It was the velocity magnitudes that were used in the simulations as velocity inlet boundary conditions, together with the x- and y-components of the flow angles.

As mentioned, the angle and Reynolds number in bold are representative of conditions at ADP of the application and will therefore be referred to as the baseline case. The inlet turbulence intensity (Tu) is set to 3.5\% and the turbulence length scale to 1.2 mm for all cases.

<table>
<thead>
<tr>
<th>Angle [deg]</th>
<th>x-component</th>
<th>y-component</th>
<th>U [m/s]</th>
<th>Re</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>****</td>
<td>****</td>
<td>****</td>
<td>2.20\times10^5</td>
</tr>
<tr>
<td>20</td>
<td>****</td>
<td>****</td>
<td>****</td>
<td>2.74\times10^5</td>
</tr>
<tr>
<td>25</td>
<td>****</td>
<td>****</td>
<td>****</td>
<td>3.20\times10^5</td>
</tr>
<tr>
<td>30</td>
<td>****</td>
<td>****</td>
<td>****</td>
<td>4.80\times10^5</td>
</tr>
</tbody>
</table>

3.2.3 Analysis Settings

ANSYS Fluent v.16.0 and ANSYS CFX v.16.0 were used in this study. As described in Chapter 2.6 there are several different transition models available in both Fluent and CFX. The settings and fluid properties for each of the models used in this thesis are listed in Appendix 1. The constants for the three transition models were left unchanged and set to default as per the recommendations of Menter et al. [10].

For the purpose of comparing the transition models with a fully turbulent model, the realizable k-\(\varepsilon\) model (rk-\(\varepsilon\)) was also used for the baseline case. The turbulence settings (turbulence intensity and length scale) used for these simulations were set the same as in the study conducted in [1]. The standard turbulence settings used was turbulence intensity (Tu) of 3.5\% and a turbulence length scale of 1.2 mm. Since it was of interest to study the effect of the turbulence intensity, the baseline case was also run with Tu = 10\%.

All Fluent simulations were run for 6000 iterations. If the convergence was questionable, the simulations were run additional iterations. The CFX simulations were run for 1000 iterations per angle as a standard approach.
3.3 Mesh Specifications

The meshes were created in ANSYS ICEM v.16.0. Transition modelling requires somewhat finer grids in order to capture all flow characteristics of the transition [10, 11]. For the purpose of studying how the mesh resolution affect the results, two different grids were used – a high-resolution mesh and a low-resolution mesh. The high-resolution mesh has approximately 11.6 million cells and the low-resolution mesh has 2.4 million cells. Both grids are of the type structured hexahedral. See Table 4 for a summary of the mesh specifications and quality.

In order to ensure a sufficient mesh quality when creating the two grids, some guidelines provided by a Design Practice at GAS [15] were followed. Menter et al. [10] and the ANSYS User Guide [11] have stated a few meshing guidelines for transition modelling:

- The streamwise grid resolution depends on the application, but at least 100 nodes from leading edge to trailing edge are required
- If separation induced transition is expected, at least 20 nodes are needed to cover the separation bubble length
- At least 30 nodes are required across the boundary layer
- Wall-normal expansion ratio (ER) ≤ 1.1
- y+ < 1

Table 4: Mesh specifications and quality. Nodes streamwise and spanwise are the number of nodes on the vane. Nodes in x- and y-direction indicate the whole domain.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Grid size</th>
<th>Nodes streamwise on vane</th>
<th>Nodes spanwise on vane</th>
<th>Nodes x-dir. in domain</th>
<th>Nodes y-dir. in domain</th>
<th>O-grid</th>
<th>ER</th>
<th>y+</th>
<th>Determinant</th>
<th>Angle</th>
<th>Volume change (max)</th>
</tr>
</thead>
<tbody>
<tr>
<td>High-Res.</td>
<td>11590158</td>
<td>200</td>
<td>200</td>
<td>320</td>
<td>90</td>
<td>1.087</td>
<td>&lt;1</td>
<td>0.90</td>
<td>29°</td>
<td>1.80</td>
<td></td>
</tr>
<tr>
<td>Low-Res.</td>
<td>2427600</td>
<td>100</td>
<td>120</td>
<td>196</td>
<td>80</td>
<td>1.23</td>
<td>&lt;1</td>
<td>0.89</td>
<td>29°</td>
<td>1.87</td>
<td></td>
</tr>
</tbody>
</table>

The guidelines for transition modelling were kept in mind when creating the high-resolution mesh and it does fulfils the ER and y+ recommendations. The y+ value is a dimensionless wall distance used in CFD. Figure 5 shows the x-wall shear stress for the baseline case with the CFX Gamma-Theta model, with the high-resolution mesh to the left and low-resolution mesh to the right. It is zoomed in to the separation bubble; the number of nodes covering the separation bubble is 19 for the high-resolution mesh and 6 for the low-resolution mesh (indicated by the blue crosses in the plots). Thus, this does not fulfil the recommendations in [10]. The number of nodes covering the separation bubble will vary from case to case, depending on the size of the separation bubble.

Figure 6 shows the velocity profiles on the vane in an effort to display the number of nodes across the boundary layer for the two meshes. For the high-resolution mesh (left in figure) the number of nodes is more than 30, which fulfils the recommendation. For the low-resolution mesh, the number of nodes is approximately 18 and thus does not fulfil the guidelines.

In order to keep the ER below 1.1 and a sufficient y+ value, the number of cells in the O-grid (wall-normal direction) of the high-resolution mesh was set to 80. The O-grid is the grid surrounding the vane closest to the surface. The first near-wall node was kept at 1.0025E-03 mm which gave a y+ value below 1 and an ER of 1.087, which is acceptable. The low-resolution mesh was modified from the high-resolution grid by reducing the number of nodes in streamwise and spanwise direction along the vane. The number of nodes in the O-grid (wall-normal direction) was
reduced as well and the first near-wall node was set at 4.0E-03 mm. This gave a $y^+$ value below 1 and an ER of 1.23. See Figure 7 for an illustration of the domain with the high-resolution mesh.

Figure 5: X-wall shear stresses for baseline case with the high-resolution mesh (left) and low-resolution mesh (right), zoomed in to area of separation. Number of nodes covering the separation bubble is 19 with the high-res mesh and 6 with the low-res mesh.

Figure 6: Velocity profiles on the vane for the baseline case with CFX Gamma-Theta model, with high-resolution mesh (left) and low-resolution mesh (right). The number of nodes across the boundary layer for the high-res mesh is more than 30, while for the low-res mesh the number of nodes is 18.
3.4 Evaluation Parameters

The most important evaluation parameter is the 2D pressure loss; although plots of the static pressure and x-wall shear stress are also presented to give more understanding of the different transition models and what transition mechanism is occurring. The axes in the diagrams and plots are normalized in order to have a meaningful comparison between different cases and models. The 2D wake profile is evaluated on a line at the plane x-out, while the 3D wake profiles are evaluated at the entire plane x-out. The static pressure and x-wall shear stress are evaluated at mid span of the vane. The mid span of the vane is shown in Figure 7. Some plots were created in Excel and some with Python or Matlab-scripts.

3.4.1 Flow Quality – Separation and Streamlines

The flow quality on the vane is determined by streamlines and flow separation in the CFD simulations. An iso-surface of the negative axial velocity is an indication of the flow separation and therefore used in the post-processing of the results. In the test data from the CTH-rig, the flow quality is determined by oil flow visualization on the vane. However, this flow visualization will not be shown in this report.

3.4.2 Vane Characteristics - Static Pressure Loadings and X-wall Shear Stress

The static pressure ($P_s$) and x-wall shear stress ($\tau_w$) are evaluated at mid span of the vane and have been normalized as follows:

\[
P_{s,\text{norm}} = \frac{P_s}{P_{s,\text{max}}} \quad (5)
\]

\[
\tau_{w,\text{norm}} = \frac{\tau_w}{\tau_{w,\text{max}}} \quad (6)
\]

And the x-wall shear stress is defined as

\[
\tau_w = \mu \frac{\partial u}{\partial y} \quad (7)
\]
where $\mu$ is the dynamic viscosity, $u$ is the flow velocity parallel to the wall and $y$ is the wall distance. A negative velocity gradient, and thus a negative wall shear stress, is an indication of separated flow.

The x-coordinate (chord length of the vane) has also been normalized in the plots according to:

$$x_{\text{norm}} = \frac{x - x_{\text{min}}}{x_{\text{max}} - x_{\text{min}}}$$  \hspace{1cm} (8)

### 3.4.3 Total Pressure Loss - 2D and 3D Wake Profiles

The evaluation of the total pressure loss has been done in several different ways; with 2D wake profile plots and integration of 2D profile pressure loss, and also with 3D wake profiles and calculation of 3D total pressure loss. The total pressure is defined as

$$p_t = p_s + q_{in}$$  \hspace{1cm} (9)

where $q_{in}$ is the dynamic pressure. The 3D pressure loss is evaluated at plane $x$-out and can be calculated in two ways:

$$P_{t,\text{loss}} = \frac{P_{t,in,mwa} - P_{t,out,mwa}}{P_{t,in,mwa}}$$  \hspace{1cm} (10)

$$P_{t,\text{loss}} = \frac{P_{t,in,mwa} - P_{t,out,mwa}}{q_{in,mwa}}$$  \hspace{1cm} (11)

The latter was used to calculate the 3D loss in this project. $P_{t,mwa}$ is the mass-weighted average total pressure at inlet or outlet. The 3D wake profiles are contour plots of the total pressure at the exit plane $x$-out and show the distribution of loss regions.

The total pressure has been normalized in the 2D wake profile plots as

$$P_{t,\text{norm}} = \frac{P_t}{P_{t,max}}$$  \hspace{1cm} (12)

where $P_{t,max}$ is the maximum total pressure on the line at the exit plane $x$-out where the 2D wake profiles have been evaluated. The 2D profile pressure loss has been calculated by integrating the 2D wake profiles and using the definition

$$P_{t,\text{loss,prof}} = \frac{P_{t,\text{out,FS}} - P_{t,out,mwa}}{q_{in,mwa}}$$  \hspace{1cm} (13)

where $P_{t,\text{out,FS}}$ is the freestream total pressure at outlet. In order to compare the different turbulence models with test data, the pressure loss difference between two different cases has been calculated using

$$P_{t,\text{loss,diff}} = \frac{P_{t,\text{loss}} - P_{t,\text{loss,ref}}}{P_{t,\text{loss,ref}}}$$  \hspace{1cm} (14)

where $P_{t,\text{loss,ref}}$ is a reference pressure loss to compare with. In this study the reference pressure loss has been from measurements in order to find out how accurate CFD is at predicting the pressure losses. These pressure loss differences will not be disclosed in this report, however.
4 Results

The results are presented by Reynolds number beginning with $2.74 \cdot 10^5$, and successively compared by mesh, turbulence model and ANSYS solver. An overview of all flow angles with Re $2.74 \cdot 10^5$ will be presented first, after which more detailed results for each inlet flow angle will be given in turn. Since the results from the simulations with the other Reynolds numbers are consistent with Re $2.74 \cdot 10^5$ (with only small differences) an overview of the results for the other Reynolds numbers will only be presented, along with the major conclusions from those simulations.

4.1 Re $2.74 \cdot 10^5$

4.1.1 Overview

Figure 8-Figure 10 shows an overview of the flow quality on the suction side of the vane for all cases, CFD tools and turbulence models run for this baseline case. All four turbulence models were used (i.e. Fluent rk-$\varepsilon$, Fluent Transition SST, CFX Gamma-Theta and CFX Gamma) together with all four inlet flow angles. The red dot in the figures indicates the cases where convergence is questionable. An overview of the flow separation and streamlines on the pressure side of the vane is available in Appendix 2.

There is an obvious trend for the rk-$\varepsilon$ model with growing separation on the endwalls with increasing inlet flow angle. The same can be said for the Fluent Transition model, with the addition of a laminar separation bubble that moves upstream with increasing flow angle. The CFX Gamma-Theta model deviates from the others at flow angle 20 degrees and shows a separation bubble on the pressure side of the vane as well.

Figure 10 shows an overview of the 3D wake profiles for all turbulence models with all four flow angles. It can be seen that the rk-$\varepsilon$ model predicts a thicker wake than any of the transition models, which is due to the turbulent boundary layer which is thicker than a laminar one. The endwall roll-up can be seen to increase with flow angle here as well.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 8: Iso-surfaces of negative axial velocity for all four turbulence models and all four flow angles (i.e. 10, 20, 25, 30 degrees). Grey areas show regions of separated flow. View from suction side of the vane. Flow direction is right to left.

Figure 9: Streamlines on the suction side of the guide vane for all four turbulence models and all four flow angles (i.e. 10, 20, 25, 30 degrees). Flow direction is right to left. Flow visualization from test has been removed.
4.1.1.1 Low-Resolution Mesh

This section will present an overview of the results from the simulations with the low-resolution mesh. It was deemed unnecessary to run the rk-ε model with the low-resolution mesh, so only the three different transition models were used.

Figure 11 and Figure 12 show an overview of the separation and streamlines on the suction side of the vane. The separation seems to follow the same trend as for the high-resolution mesh, although with a less well-defined laminar separation bubble. The Fluent simulations were less inclined to converge with the low-resolution mesh compared to the simulations with the high-resolution mesh.

The CFX Gamma-Theta model deviates at flow angle 20 and 25 degrees from the other two transition models by showing separation on the pressure side of the vane. The figures of the separation and streamlines on the pressure side of the vane can be seen in Appendix 2.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 11: Iso-surfaces of negative axial velocity for all three transition models and all four flow angles (i.e. 10, 20, 25, 30 degrees). Grey areas show regions of separated flow. View from suction side of the vane. Flow direction is right to left.

Figure 12: Streamlines on the suction side of the guide vane for the three transition models. Flow direction is right to left. Flow visualization from test has been removed.

4.1.2 10 Degrees

Figure 13 shows contour plots of the total pressure at x-out (i.e. the 3D wake profiles) and it shows that the Fluent Transition model has the thinnest wake profile with either mesh. Red indicates high total pressure and high velocity. Figure 14 shows plots of the 2D wake profiles for all three transition models. There is not much difference between the transition models; only the CFX Gamma-Theta model displays a slightly deeper wake (with both meshes). A deeper and wider wake essentially means a larger pressure loss.

The static pressure loadings in Figure 15 show very similar load curves for the transition models, the only difference is the laminar separation bubble present for the Fluent Transition model and CFX Gamma-Theta model which the CFX Gamma model does not predict. The separation bubble shows up as a plateau in the load curves and is circled in the figure.

The plot of the x-wall shear stress in Figure 16 shows that the CFX Gamma model triggers an earlier transition than the other two models. The CFX Gamma model starts transition at approximately 50% of the chord, while it starts at 55% and 57% of the chord for the Fluent
Transition model and CFX Gamma-Theta model respectively. The Fluent Transition model and the CFX Gamma-Theta model give quite a similar result, though a larger laminar separation bubble is predicted with the Gamma-Theta model. The separation bubble also begins somewhat downstream with the Gamma-Theta model compared with the Fluent Transition model; but the transition is completed at approximately 65% of the chord for both models with the high-resolution mesh. With the low-resolution mesh, the Fluent Transition model and the CFX Gamma-Theta model begin transition at the same point (at 58% of the chord) though the former completes the transition to turbulent faster. All plots with the low-resolution mesh are available in Appendix 2.

Also seen in Figure 16 is that while the CFX Gamma model predicts a bypass transition, the Fluent Transition model and the CFX Gamma-Theta model show a separation induced transition (indicated by the negative wall shear stress). All three transition models predict separation induced transition on the pressure side of the vane, however.

Figure 13: The 3D wake profiles evaluated at plane x-out for the three transition models. High-resolution mesh (top) and low-resolution mesh (bottom). The scale has been removed.

Figure 14: Plot of the normalized wake profiles with the different transition models and the high-resolution mesh.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

4.1.2.1 Comparison with Test Data – 10 Degrees

This section will compare the results for the Fluent Transition model (high-resolution mesh only) with the test data, although no pictures or plots will be shown. The streamlines are compared with the flow visualization as described in section 3.4 and they show good agreement between test and CFD in terms of laminar separation and endwall roll up.

The 3D wake profiles are very similar, although the wake seems slightly thicker for CFD while the test data shows a thicker boundary layer at the endwalls. CFD shows good agreement in the depth of the 2D wake profiles but predicts a slightly thinner wake than the test data shows. CFD also have some trouble predicting the position of the wake, though it could be that the wake in the test rig is not completely stationary.

CFD shows very good agreement with test data in the static pressure load curves, with the separation bubble located at approximately 50% of the chord for both CFD and test data.

4.1.3 20 Degrees (ADP)

This case is representative of the ADP in an engine application and was therefore more extensively simulated than the other cases. The results for the simulations with the original turbulence intensity of 3.5% are presented first, and then the results from the investigation with a higher turbulence intensity that was performed for this case.
The CFX Gamma-Theta and CFX Gamma simulations were not properly converged for either mesh, thus the results might be unreliable.

An overview of the flow separation for angle 20 degrees on the suction side and pressure side of the vane with the high-resolution mesh and low-resolution mesh side by side can be seen in Appendix 2.

The 3D wake profiles in Figure 17 shows that the CFX Gamma model predicts larger endwall roll up while the CFX Gamma-Theta model has the thickest wake. The red indicates regions of high velocity and high total pressure, while the blue indicates regions of low velocity and low total pressure. In Figure 18 the 2D wake profiles for the Fluent Transition model and the CFX Gamma model are almost identical while the CFX Gamma-Theta model predicts a wider wake. In the 2D wake profile plots with the low-resolution mesh the three transition models are more similar to each other compared with the high-resolution mesh. These plots with the low-resolution mesh are shown in Appendix 2.

Figure 19 shows the static pressure loadings on the vane for the three transition models. The load curves are very similar between the two meshes. The Fluent Transition model and the CFX Gamma-Theta model once more predict a separation bubble (plateau in the load curve) that the Gamma model does not. Also seen in Figure 19 is that the Gamma model shows slightly smaller loading on the suction side of the vane compared to the other two models.

Figure 20 shows plots of the x-wall shear stress. Similar to the previous case (flow angle 10 degrees), the transition starts and ends further upstream with the Gamma model; it begins at approximately 44% of the chord for the Gamma model, while it begins at 50% of the chord for the other two transition models. Both the Fluent Transition model and the CFX Gamma-Theta model show flow separation before transition occurs; this is indicated by the negative x-wall shear stress in Figure 20. The shear stress for the Gamma model never reaches a negative value before going into transition, which is an indication of bypass transition.

![Figure 17](image-url)
4.1.4 Turbulence Intensity Investigation Fluent - 20 degrees

In order to determine how the turbulence intensity level (Tu) affect transition, the baseline case was run with both Tu = 3.5% and Tu = 10%. This was done for the high-resolution mesh only. Figure 21 shows a comparison of the separation, streamlines and 3D wake profiles between the two cases. The higher turbulence intensity results in smaller endwall separation and roll-up. No laminar separation bubble is present on the vane when Tu = 10%; a higher turbulence intensity means that bypass transition is most likely the transition mechanism occurring. This is also
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

apparent from the plot of x-wall shear stress on the vane in Figure 22. The transition occurs more gradually and further upstream when the turbulence intensity is high. In addition, the x-wall shear stress shows that transition occurs without reaching a negative value; this is an indication of bypass transition, as described in Chapter 2.2.

The 2D wake profiles in Figure 22 are quite similar between the two cases however, although Tu = 3.5% shows a slightly deeper wake. Figure 23 shows a comparison of the static pressure loading on the vane, where the lack of separation bubble is once again evident for the case with higher turbulence intensity.

![Figure 21: Iso-surfaces of negative axial velocity, streamlines and 3D wake profiles (in Pa) for Re \(2.74 \cdot 10^5\) and flow angle 20 degrees with turbulence intensities 3.5% (left) and 10% (right).](image)

![Figure 22: Plot of x-wall shear stress (left) on the guide vane, and the 2D wake profiles (right) with different turbulence intensities.](image)
4.1.4.1 Comparison with Test Data – 20 Degrees

This section will present a comparison between the results from the Fluent Transition model and test data for flow angle 20 degrees, although no pictures or plots from the test data will be shown. The streamlines and the flow visualization show good agreement in terms of the separation bubble. The streamlines show a slightly more curved separation bubble however, but this is could be caused by the larger roll-up at the endwalls which increases the flow velocity on the vane.

The 3D wake profile for the CFD simulations shows larger endwall roll-up and a more curved wake, while the test data shows a thicker boundary layer at the endwalls. The 2D wake profiles are similar in depth but slightly thinner for the CFD simulations. The static pressure load curves show very good agreement between test and CFD.

4.1.5 25 Degrees

An overview of the flow separation on the suction and pressure side of the vane for both meshes side by side is available in Appendix 2. In that overview it is evident the separation bubble predicted with the low-resolution mesh for the Fluent Transition model is less well-defined than the separation bubble predicted with the high-resolution mesh. The CFX Gamma-Theta and CFX Gamma models are once again not well-converged.

Figure 24 shows the 3D wake profiles for all three transition models and both meshes. With the high-resolution mesh, the CFX Gamma-Theta model shows a large separation at mid span of the vane which is evident by the thick wake profile.

The 2D wake profile for the CFX Gamma-Theta model in Figure 25 is much larger than for the other two models due to the large separation occurring at mid span of the vane. That separation is also evident in the plot of the x-wall shear stress in Figure 27; at approximately 70% of the chord length, the x-wall shear stress is negative until reaching the trailing edge of the vane.

The static pressure loading in Figure 26 are quite similar between the transition models although the CFX Gamma model differs once more by not predicting a separation bubble.

In Figure 27 of the x-wall shear stress it can be seen that the transition predicted with the CFX Gamma model is more gradual and starts further upstream on the vane compared to the Fluent Transition model. For the Gamma model, transition begins at approximately 40% of the chord while it starts at 46% for the Fluent Transition model for both meshes. The x-wall shear stress for the Fluent Transition model and CFX Gamma-Theta model reaches negative values, thus
indicating separation induced transition. All corresponding plots with the low-resolution mesh are available in Appendix 2.

Figure 24: The 3D wake profiles for flow angle 25 degrees, for the three transition models with the high-resolution mesh (top) and low-resolution mesh (bottom).

Figure 25: Plot of the normalized wake profiles with the three transition models for flow angle 25 degrees and the high-resolution mesh.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 26: Normalized static pressure on the vane with the three transition models for flow angle 25 degrees and the high-resolution mesh.

Figure 27: Comparison of the x-wall shear stress on the vane with the three transition models for flow angle 25 degrees and the high-resolution mesh.

4.1.5.1 Comparison with Test Data - 25 degrees

This section will present a comparison of the results from the Fluent Transition model with the test data for flow angle 25 degrees, although no pictures or plots from test data will be shown.

The streamlines from CFD and the flow visualization from test both display a laminar separation and endwall roll-up. The streamlines show a slightly more curved separation bubble however, but this is could be caused by the larger roll-up at the endwalls which increases the velocity of the flow on the vane.

The 2D wake profile for the Fluent Transition model is deeper but thinner than the test data. The static pressure loading shows a similarity between test data and CFD, although the suction peak for CFD is somewhat deeper compared to test. This could mean that the flow velocity on the vane is higher in CFD.

4.1.6 30 degrees

An overview of the flow separation on the suction side and pressure side of the vane with both meshes side by side is available in Appendix 2. The results for the Fluent Transition model and the CFX Gamma-Theta model with the high-resolution mesh are very similar in terms of endwall separation and laminar separation bubble. The Fluent Transition model shows good agreement between the two meshes; the difference is the separation bubble which is not well-defined with the low-resolution mesh. The CFX simulations might have unreliable results since their convergence is questionable.
The wake contours in Figure 28 for all three models are very similar (with the high-resolution mesh), though the wake for CFX Gamma-Theta is slightly thicker. This is also noticeable in the 2D wake profile plots in Figure 29 where the Gamma-Theta model displays a wider wake. The red in the wake contours indicate regions of high flow velocity and high total pressure, while blue indicates regions of low velocity low total pressure.

Figure 30 of the static pressure loading on the vane shows similar results for the Fluent Transition model and the CFX Gamma-Theta model, while the CFX Gamma model differs and shows no separation bubble.

The Gamma model is continuing the trend of an earlier and more gradual transition; the x-wall shear stress in Figure 31 shows that the Gamma model transitions without the x-wall shear stress reaching negative values, which is an indication of bypass transition. The Gamma model starts transition at approximately 36% of the chord, while for the Fluent Transition model and CFX Gamma-Theta model transition starts at 42% and 43% of the chord respectively. Both the latter models show a separation induced transition. All corresponding plots with the low-resolution mesh are shown in Appendix 2.

Figure 28: The 3D wake profiles for flow angle 30 degrees evaluated at plane x-out, for the three transition models. High-resolution mesh (top) and low-resolution mesh (bottom).
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

4.1.6.1 Comparison with Test Data – 30 Degrees

This section will compare the results from the Fluent Transition model with test data for flow angle 30 degrees, although no pictures or plots from test data will be shown.

The endwall roll-up is once again larger for the CFD compared to the test data, but the streamlines and flow visualization show good agreement in terms of the separation bubble. The 3D wake profile is thicker for CFD, which is also evident in the 2D wake profiles where the Fluent Transition model predicts a wider and much deeper wake.
Similar to the previous case (flow angle 25 degrees), CFD predicts a deeper suction peak in the static pressure loading compared to test data. This could be a result of higher flow velocity on the vane in CFD.

4.1.7 **Comparison With rk-ε Model – Re 2.74\cdot10^5**

This section will show a comparison between the Fluent Transition model and the fully turbulent model rk-ε; first an overview and then a slightly more detailed look at flow angle 20 degrees. Only the high-resolution mesh was used in this comparison.

Figure 32 shows the flow separation on the suction side of the vane for both models and it can be seen that the fully turbulent model does not predict the laminar separation that the test data shows.

![Figure 32: Iso-surface of negative axial velocity on suction side of the vane. Grey areas show regions of separated flow. Comparison between rk-ε and Fluent Transition model of all four flow angles. Flow direction is right to left.](image)

4.1.7.1 **Comparison with rk-ε Model – 20 Degrees**

All four flow angles were used with the rk-ε model, but only a closer look at the results for flow angle 20 degrees will be presented here. The 2D wake profile is significantly larger with a fully turbulent model compared to a transition model, see Figure 33. The x-wall shear stress in Figure 33 is higher for the turbulent model, as is expected, and also shows that the fully turbulent model does not predict a transition.

Figure 34 is a comparison of the static pressure loading on the vane between the two models and it shows that the fully turbulent model does not capture the separation bubble like the transition models does.

The 3D wake profile in Figure 35 is also significantly thicker for the fully turbulent model compared to the transition model, which is to be expected since a turbulent boundary layer is thicker than a laminar one.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 33: Comparison of wake profiles (left) and x-wall shear stress (right) between the Fluent Transition and rk-ε models for flow angle 20 degrees.

Figure 34: Comparison of static pressure loadings between the Fluent Transition and rk-ε models for flow angle 20 degrees.

Figure 35: 3D wake profiles comparison between the Fluent Transition model (left) and rk-ε model (right) for inlet flow angle 20 degrees.

4.1.8 Conclusions Re 2.74×10^5

The CFX Gamma model consistently predicts an earlier and more gradual transition than either the Fluent Transition model or the CFX Gamma-Theta model. All three transition models show the same trend in that the laminar separation bubble and transition point moves upstream with increasing inlet flow angle.
The agreement in the results between the high- and low-resolution mesh (for the Fluent simulations) is very good. The difference is that the low-resolution mesh could not fully resolve the laminar separation bubble and thus had some convergence issues. The two CFX transition models show more differences between the results with the two meshes, although they also had more convergence issues initially.

The CFD simulations consistently predict larger endwall roll-up of the boundary layer compared to test data. However, the CFD simulations and the test data show good agreement in terms of the vane static pressure loadings and the location of the separation bubble.

### 4.2 Re $2.20 \cdot 10^5$

This section will present an overview of the flow quality on the vane with the high-resolution mesh and only the major conclusions from the studies with Re $2.20 \cdot 10^5$. The low-resolution mesh was only used for flow angle 20 degrees with this Reynolds number, though it is not displayed here.

#### 4.2.1 Overview

Figure 36 and Figure 37 shows an overview of the streamlines and flow separation for all inlet flow angles and transition models used with the high-resolution mesh. The red dot indicates the cases where convergence is questionable. Overviews of the separation and streamline on the pressure side of the vane are available in Appendix 2.

The Fluent Transition model and CFX Gamma-Theta model show very similar endwall separation and roll-up, except for the separation bubble on the pressure side for the latter (see Appendix 2). Unlike the simulations with Re $2.74 \cdot 10^5$ however, the simulations with Re $2.20 \cdot 10^5$ show a slightly larger separation bubble for the CFX Gamma model. This is to be expected, since a laminar separation bubble grows with decreasing Reynolds number, as described in section 2.4.3.

![Figure 36](image)

**Figure 36:** Iso-surfaces of negative axial velocity for all three transition models and all four flow angles (i.e. 10, 20, 25, 30 degrees). Grey areas show regions of separated flow. View from suction side of the vane. Flow direction is right to left.
4.2.2 Major Conclusions Re $2.20 \cdot 10^5$

The CFX Gamma model continuously shows separation induced transition for this Reynolds number, unlike the simulations with Re $2.74 \cdot 10^5$.

The results for the high- and low-resolution mesh show good agreement, especially in terms of the wake profiles. It is apparent that the low-resolution mesh cannot capture the entire separation bubble however. The low-resolution mesh also had convergence issues once again, which is probably a result of the unsteady separation bubble.

CFD predictions and test data generally show good agreement in terms of wake profiles and static pressure loadings.

4.3 Re $3.20 \cdot 10^5$

This section will present an overview of the flow quality on the vane with the high-resolution mesh and the major conclusions from the studies with Re $3.20 \cdot 10^5$. The low-resolution mesh was only used for flow angle 20 and 25 degrees with this Reynolds number, though it is not displayed here.

4.3.1 Overview

Figure 38 and Figure 39 shows an overview of the flow quality on the vane for the performed simulations with the high-resolution mesh. The red dot indicates the cases with unreliable results due to convergence issues. In line with previous simulations, the endwall separation grows with increasing flow angle. The CFX Gamma-Theta model once again predicts a small separation bubble on the pressure side of the vane for flow angle 20 degrees that the other two models do not show. See Appendix 2 for figures of the separation and streamlines on the pressure side of the vane.

Figure 37: Streamlines on the suction side of the guide vane for all three transition models with the high-resolution mesh. Flow direction is right to left. Flow visualization from the test rig has been removed.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 38: Iso-surfaces of negative axial velocity for all three transition models and all four flow angles (i.e. 10, 20, 25, 30 degrees). Grey areas show regions of separated flow. View from suction side. Flow direction is right to left.

Figure 39: Overview of streamlines on the suction side of the vane. Flow direction is right to left. Flow visualization from the test rig has been removed.

4.3.2 Major Conclusions Re 3.20·10^5

Unlike for Re 2.20·10^5, these simulations with the CFX Gamma model consistently predict bypass transition while the Fluent Transition model and CFX Gamma-Theta model once again display separation induced transition. This is indicated by the x-wall shear stress where the Gamma model never reaches negative values before transition, while the other two models indicate separation before going into transition.

CFD and test data show quite good agreement here in terms of the 2D wake profiles and the static pressure loadings. CFD predicts larger endwall roll-up, however.

4.4 Re 4.80·10^5

This section will present an overview of the flow quality on the vane with the high-resolution mesh and the major conclusions from the studies with Re 4.80·10^5. The low-resolution mesh was only used for flow angle 20 degrees with this Reynolds number, though it is not displayed here.
4.4.1 Overview

Figure 40 and Figure 41 show an overview of the flow quality on the suction side of the vane for the high-resolution mesh. The red dot indicates the cases with unreliable results due to convergence issues. Overviews of the separation and streamlines on the pressure side of the vane are available in Appendix 2. In Figure 40 it can be seen that the endwall separation and the separation bubble is smaller and thinner than for previous Reynolds numbers. This is because the boundary layer grows thinner when the Reynolds number increases.

![Figure 40: Iso-surfaces of negative axial velocity for all three transition models and all flow angles (i.e. 10, 20, 25, 30) with the high-resolution mesh. Grey areas show regions of flow separation. View from suction side of the vane. Flow direction is right to left.](image)

![Figure 41: Overview of streamlines on the suction side of the vane. Flow visualization from the test rig has been removed. Flow direction is right to left.](image)

4.4.2 Major Conclusions Re 4.80·10⁵

The CFX Gamma model consistently displays bypass transition with this Reynolds number, whereas the Fluent Transition model and CFX Gamma-Theta model display separation induced transition.
The test data do not show a laminar separation like the Fluent Transition model and CFX Gamma-Theta model predict. CFD predicts similar depth in the 2D wake profiles, although thinner than test data.
4.5 Loss Predictions

In this section the results from the calculations of the pressure losses will be presented in tables and figures. Table 5 shows the 3D losses for the simulations with the high-resolution mesh and Table 6 and shows the integrated 2D losses for the high-resolution mesh. In the tables, “Fluent” indicates the Fluent Transition model. The test data has been removed from the tables. Corresponding tables for the calculated 2D and 3D pressure losses with the low-resolution mesh are shown in Appendix 2.

**Table 5: 3D-losses (dP/t,q,in) for all turbulence models, CFD codes, Reynolds numbers and flow angles with the high-resolution mesh. The test data has been removed.**

<table>
<thead>
<tr>
<th>Re</th>
<th>Angle</th>
<th>rk-ε</th>
<th>Fluent</th>
<th>CFX-GT</th>
<th>CFX-G</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.20·10^5</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>5.42%</td>
<td>5.65%</td>
<td>6.13%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>6.66%</td>
<td>6.86%</td>
<td>6.84%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.74·10^5</td>
<td>10</td>
<td>7.05%</td>
<td>3.62%</td>
<td>3.76%</td>
<td>3.70%</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>7.53%</td>
<td>4.75%</td>
<td>3.99%</td>
<td>5.70%</td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>8.40%</td>
<td>5.97%</td>
<td>4.96%</td>
<td>7.74%</td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>9.29%</td>
<td>7.12%</td>
<td>9.45%</td>
<td>7.73%</td>
</tr>
<tr>
<td>3.20·10^5</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>4.17%</td>
<td>3.66%</td>
<td>3.43%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>5.25%</td>
<td>3.79%</td>
<td>5.27%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.80·10^5</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>3.72%</td>
<td>3.75%</td>
<td>3.38%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>5.17%</td>
<td>5.13%</td>
<td>4.76%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 6: Integrated 2D-losses for all turbulence models, CFD codes, Reynolds numbers and flow angles with the high-resolution mesh. The test data has been removed.

<table>
<thead>
<tr>
<th>Re</th>
<th>Angle</th>
<th>rk-ε</th>
<th>Fluent</th>
<th>CFX-GT</th>
<th>CFX-G</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.20·10^5</td>
<td>10</td>
<td>2.08%</td>
<td>2.42%</td>
<td>2.06%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>2.63%</td>
<td>3.12%</td>
<td>2.69%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.74·10^5</td>
<td>10</td>
<td>3.35%</td>
<td>1.81%</td>
<td>1.92%</td>
<td>1.88%</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>3.54%</td>
<td>1.80%</td>
<td>2.28%</td>
<td>1.72%</td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>3.82%</td>
<td>2.14%</td>
<td>4.45%</td>
<td>2.30%</td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>4.54%</td>
<td>3.29%</td>
<td>3.50%</td>
<td>2.86%</td>
</tr>
<tr>
<td>3.20·10^5</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>1.68%</td>
<td>1.81%</td>
<td>1.78%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>1.83%</td>
<td>1.92%</td>
<td>1.97%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.80·10^5</td>
<td>10</td>
<td>1.50%</td>
<td>1.57%</td>
<td>1.67%</td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>1.56%</td>
<td>1.63%</td>
<td>1.79%</td>
<td></td>
</tr>
</tbody>
</table>

The following figures show the pressure losses in percent for all turbulence models plotted together for comparison. The pressure losses from the test data have been left out in all plots but will be referred to in general terms. The 3D losses for Re 2.74·10^5 are presented first and then the 3D losses for flow angle 20 degrees. After that, the 2D losses are presented in the same way; first for Re 2.74·10^5 and then for flow angle 20 degrees.

All turbulence models with both high- and low-resolution mesh is plotted in Figure 42. The dotted lines show the pressure losses with the low-resolution mesh. Also included are the pressure loss from the simulations with Tu = 10%. Take into account that the CFX results are unreliable (except for cases with flow angle 10 degrees) due to convergence issues.

The fully turbulent model rk-ε (light blue line) over predict the 3D losses while the transition models under predict the pressure losses at lower angles compared to test data. At flow angles 25 and 30 degrees, the Fluent Transition model (blue line) predicts the 3D losses very well. It can also be seen in Figure 42 that there is very good agreement between the high- and low-resolution mesh for the simulations with the Fluent Transition model. This will be further discussed in Chapter 5.4. Meanwhile, the agreement is not as good between the two meshes for the CFX transition models.
The 3D pressure losses for flow angle 20 and all four Reynolds numbers are plotted in Figure 43. Test data has been removed from the graph. CFD (the transition models) under predicts the 3D losses compared to test data. The CFX transition models and the rk-ε model were only used for Re \(2.74 \times 10^5\) and are therefore shown simply as one point in the graph.

Figure 44 shows the 2D pressure losses for all turbulence models with Re \(2.74 \times 10^5\) and all four flow angles. The test data has been removed. The results from the simulations with the low-resolution mesh are also plotted (dotted lines). Also included is the result with Tu = 10%.

The rk-ε model (light blue line) greatly over predicts the 2D losses while the Fluent Transition model shows good agreement with test data and follows the same trend for flow angles 10, 20 and 25 degrees. At 30 degrees, all transition models over predict the 2D losses. The steep gradient of the curve between 25 and 30 degrees (for the Fluent Transition model) could be due to the increase in loading on the vane as the endwall separation increases. The “peak” in the loss at flow angle 25 degrees for the CFX Gamma-Theta model (red line) is a result of the large separation occurring at mid span of the vane, which can be seen in Figure 8.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 44: Plot of the integrated 2D losses with both meshes and all turbulence models for Re \(2.74 \times 10^5\) and all four flow angles (i.e. 10, 20, 25, 30). Test data has been removed.

Figure 45 shows the 2D pressure losses for flow angle 20 degrees for all four Reynolds numbers. The \(r\kappa-\varepsilon\) model is included, as well as results from both meshes for all the transition models. In Figure 45, the transition models show good agreement with each other and with test data (same trend and only slightly under predicting the losses).

Overall, the 2D and 3D losses increase with increasing flow angle and decrease (or level out) with increasing Reynolds number.

Figure 45: Plot of the 2D pressure losses for all turbulence models for flow angle 20 degrees and all four Reynolds numbers. Test data has been removed.
5 Discussion

This chapter will aim to answer the research questions posed in section 1.3.

5.1 Is the Flow Transitional in the Test Rig?

Indicated by the flow visualization on the vanes (not included in this report), the flow is transitional in the test rig. The static pressure loadings also show laminar separations.

5.2 Which Transition Model or CFD Tool Works Best?

The CFX simulations constantly had more convergence issues than Fluent, even when using smaller and/or larger time scales. If getting CFX to converge is deemed to be important, it could be something to look into further; for example running transient simulations instead of steady state to see if CFX can reach convergence. The fact that the CFX results are unreliable makes it difficult to compare with the Fluent simulations. However, the Fluent Transition model and CFX Gamma-Theta model does show a similarity in the results, which would perhaps be expected since it is the same transition model only implemented in different CFD tools.

The fact that the CFX Gamma model consistently displays transition through bypass, with the exception of the lowest Reynolds number, could be due to the default constant value of $Re_\theta = 260.0$ implemented in the model. According to Langtry [8]: "if a constant transition momentum thickness Reynolds number is specified, the transition model is not very sensitive to adverse pressure gradients". And seeing as it is the adverse pressure gradient that cause separation induced transition, it might mean that the CFX Gamma model needs to be tuned, specifically the $Re_\theta$ value. How that should be done, what variable or function to set instead of 260.0, is unclear and would need more extensive studies of this transition model.

Overall, the Fluent Transition model works best, since it had less convergence issues than CFX.

5.3 How Accurate is CFD in Comparison to Test Data?

In order to compare CFD with test data the total pressure loss difference was calculated, as described in section 3.4.3, and tabulated as an addition to the graphs in the chapter of loss predictions (section 4.5). These tables are not disclosed in this report however, due to confidentiality, but the results will be discussed in general terms.

The Fluent Transition model shows quite good agreement with test data when predicting the 2D pressure losses. The transition model has more difficulty in predicting the 3D losses. Reasons for the transition models inability to predict the 3D losses well could be due to the boundary layer at the endwalls; in the test rig there already exists a boundary layer which has a certain thickness, while there is no boundary layer in the CFD simulations initially. A large part of the 3D pressure losses might originate from the endwalls which the rk-$\varepsilon$ model is better at capturing than the Fluent Transition model. The rk-$\varepsilon$ model greatly over predicts the 2D losses however. The CFX transition models sometimes show better agreement with test data than the Fluent Transition model but since they are not well converged, these loss predictions are unreliable.

There are several possible reasons for the discrepancies between CFD and test data. First of all, there are differences between the two vanes in the test rig due to the rig not being completely symmetrical. The airflow over the two vanes can differ in both velocity and flow angle and thus result in different loadings and pressure losses. Secondly, the flow conditions in the CFD simulations were not the same as in the test rig. The velocity in the rig was not exactly as it was in the simulations and the flow angle was not exactly as it was in the simulations. i.e. tuning of the CFD simulations might be necessary to get a better agreement.
5.3.1 How Does Transition Vary With Flow Angle and Reynolds Number?

The laminar separation bubble, and thus the transition point, moves upstream with increasing flow angle. This is demonstrated in Figure 46 of the wake profiles, load curves and x-wall shear stress for the Fluent Transition model with Re $2.74 \times 10^5$ and all four flow angles. In the plot of the x-wall shear stress it can be seen that the transition point moves from approximately 56% of the chord (at 10 degrees) to 41% of the chord (at 30 degrees). The wake profiles grow larger and the loading on the vane also becomes larger with increasing flow angle, as would be expected since the endwall separation also increases (see Figure 48).

In Figure 47 it can be seen that the transition point is not much affected by the increasing Reynolds number (see plot of x-wall shear stress). Figure 49 also shows how the endwall separation and the laminar separation bubble become smaller with increasing Reynolds number, due to the decreasing thickness of the boundary layer.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 49: Flow angle 20 degrees with increasing Re, for the Fluent Transition model. Separation decreases with increasing Re.

5.4 How Does the Mesh Resolution Affect the Results?

The 2D and 3D pressure losses do not differ much between the high- and low-resolution mesh. The largest difference seems to occur at high flow angles (see Figure 50). However, if a laminar separation induced transition is occurring, the low resolution mesh cannot fully capture the separation bubble. This is evident in the plots of the x-wall shear stress and load curves presented in Appendix 2, where the separation bubble appears to be unsteady. There is also an issue with convergence for the simulations with the low-resolution mesh, which is most likely due to the unsteady laminar separation bubble. In conclusion, if a laminar separation induced transition is predicted and there are convergence issues the mesh might need to be refined. Additional plots with flow angle 20 degrees and all Reynolds numbers are available in Appendix 2.

Figure 50: Comparison of calculated 3D losses (left) and 2D losses (right) between high- and low-resolution mesh for Re 2.74-10^5.

5.5 How Does the Turbulence Intensity Affect Transition?

The transition mechanism is sensitive to the turbulence intensity level; when the turbulence intensity is high, the transition occurs through bypass, as opposed to separation induced transition when the turbulence intensity is lower. The endwall separation is larger with lower turbulence intensity, but the difference in the total pressure loss is not overly large between the two cases, as can be seen in Figure 51. The case with the higher turbulence intensity (Tu=10 %) is shown in green in the plots.
5.6 What Should a Best Practise be for Transition Modelling?

A best practise for transition modelling for this application should be to use Fluent Transition model. The mesh created according to current best practise is sufficient for predicting same pressure losses as a high-resolution mesh created with guidelines for transition modelling. However, if a laminar separation bubble is predicted the low-resolution mesh might need to be locally refined for better convergence.
6 Conclusions

This chapter will present a short summary of the thesis work. The findings made during this study will also be listed together with a summary of the conclusions from the Results chapter and the Discussion chapter.

In this master’s thesis, several cases were studied with varying Reynolds numbers, flow angles and turbulence intensities in order to investigate aerodynamic performance parameters like flow separation and pressure loss for an OGV. This was accomplished by applying three different transition models in ANSYS Fluent and CFX. Two meshes with different resolutions were also used. It was of interest to compare a fully turbulent model with the transition models, therefore the Fluent rk-ε model was applied to the baseline case in this study.

It was found that the Fluent Transition model works better than the CFX transition models due to convergence issues with CFX. It was also found that the transition models show good agreement with test data in terms of static pressure loadings on the vane, wake pressure profiles and 2D pressure losses. Furthermore, the Fluent Transition model predicts a laminar separation induced transition like the test data shows (for all cases except the highest Reynolds number).

6.1 Summarized Findings

CFD vs. Test data:
- CFD (Fluent Transition model) consistently predict separation induced transition compared to test data
- The transition models under predict 3D losses, while rk-ε model is conservative (i.e. over predict)
- The transition models show quite good agreement with test for 2D losses, while rk-ε model is overly conservative

Fluent vs. CFX:
- CFX generally has more convergence issues
- CFX Gamma model consistently triggers transition further upstream compared to the other two transition models

Mesh resolution:
- The separation bubble cannot be fully resolved with the low-resolution mesh
- There were convergence issues with the low-resolution mesh, most likely due to the unsteady separation bubble. The mesh might need refinement if this is predicted
- The differences in the pressure losses between the high- and low-resolution mesh are very small (for Fluent simulations)

6.2 Summarized Conclusions

Which transition model or CFD tool works best, in general?
- The Fluent Transition SST model work best due to the instability of CFX

How accurate is CFD in comparison to test data?
- The transition models are more accurate than a fully turbulent model at capturing the physics of the transition phenomenon
The transition models show good agreement when predicting the 2D losses.
The fully turbulent model greatly over predicts the 2D losses.
The transition models under predict the 3D losses.

How does transition vary with flow angle and Re?
- The transition point moves upstream with increasing flow angle.
- The laminar separation bubble grows thinner and the endwall separation decreases with increasing Reynolds number.
- The transition point is not much affected by increasing Reynolds number.

How does the turbulence intensity affect transition?
- A higher Tu results in bypass transition, but the difference in pressure losses is very small.

How does the mesh resolution affect the results?
- A low-resolution mesh cannot capture the entire separation bubble which results in convergence issues.
- The difference in pressure losses between a high- and low-resolution mesh is very small.

What should a best practice be for transition modelling for this application?
- Run Fluent Transition SST model.
- Current best practice for meshing is sufficient, if there are no convergence issues.
7 Suggested Future work

This chapter will present some suggested future work that can be done in order to take the next step in the validation of the transition models.

First of all, to use post-test inlet boundary conditions as profiles (instead of simply an inlet velocity magnitude as for this study) and better matched to the conditions in the test rig. A more comprehensive mesh study could also be done, in order to find a truly mesh independent solution and to investigate whether it is the nodes in the O-grid or the streamwise or spanwise number of nodes that are important in resolving the separation bubble. A deeper understanding of the transition phenomenon could be gained by calculating the boundary layer velocity profiles and studying the boundary layer development during transition. It would also be interesting to extend the study surface roughness, to see how the transition is affected and how well the transition model predicts that. And finally, to extend the study to an engine application with realistic geometry and more engine realistic operating conditions.
8 References


[4] Ström, L.: Test specification for aero testing of impact of surface characteristics on aero on aero performance in the large scale, low speed linear cascade rig at Chalmers, Reg nr. VOLS:10211343


[13] Nichols, R.H., Turbulence Models and Their Application to Complex Flows, Revision 4.01, University of Alabama at Birmingham


Appendix 1: Material Properties and Turbulence Models Setup

Fluid properties and model constants used for the simulations are described in the following list.

Fluid/Air:
- Density = 1.225 k/m$^3$
- Specific heat (Cp) = 1006.43 J/kg K
- Thermal conductivity = 0.0242 W/m K
- Viscosity = 1.7894e-05 kg/m s
- Molecular weight = 28.966 kg/kmol
- Thermal expansion coefficient = 0 K$^{-1}$

Boundary conditions at walls:
- No slip walls

Model constants for the intermittency:
- Default settings

Model constant for the transition onset momentum-thickness Re ($Re_{th}$):
- Default settings
Appendix 2: Additional Results

Re 2.74·10⁵ – Flow Quality Overview, High-Resolution Mesh

Figure 52 shows an overview of the flow separation on the pressure side of the vane, for Re 2.74·10⁵ and the high-resolution mesh. Figure 53 shows the streamlines on the pressure side of the vane in the same way.

Figure 52: Iso-surfaces of negative axial velocity for all four turbulence models and all four flow angles (i.e. 10, 20, 25, 30 degrees) with the high-resolution mesh. Grey areas show regions of separated flow. View from pressure side of the vane. Flow direction is left to right.

Figure 53: Streamlines on the pressure side of the guide vane for all four turbulence models and all four flow angles (i.e. 10, 20, 25, 30 degrees) with the high-resolution mesh. Flow direction is left to right. Flow visualization from test has been removed.
**Re 2.74 \cdot 10^5 – Flow Quality Overview, Low-Resolution Mesh**

Figure 54 and Figure 55 shows the flow separation and streamlines on the pressure side of the vane for Re $2.74 \cdot 10^5$ and all flow angles with the low-resolution mesh. The rk-ε model was not used with this mesh, thus only the three transition models are shown.

![Figure 54](image1)

**Figure 54**: Iso-surfaces of negative axial velocity for all three transition models and all four flow angles (i.e. 10, 20, 25, 30 degrees) with the low-resolution mesh. Grey areas show regions of separated flow. View from pressure side of the vane. Flow direction is left to right.

![Figure 55](image2)

**Figure 55**: Streamlines on the pressure side of the vane for the three transition models. Flow direction is left to right.

**Re 2.74 \cdot 10^5 – 10 Degrees**

Figure 56 shows a comparison of the separation between the high- and low-resolution mesh for flow angle 10 degrees. For the low-resolution mesh, the laminar separation bubble seems less well-defined with the Fluent Transition model compared with the high-resolution mesh. Figure 57- Figure 59 shows plots of the 2D wake profiles, the static pressure load curves and the x-wall shear stress for the low-resolution mesh.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 56: Iso-surfaces of negative axial velocity on the suction side (SS) and pressure side (PS) of the vane. Flow angle is 10 degrees. Grey areas indicate regions of separated flow. High-resolution mesh (left) and low-resolution mesh (right).

Figure 57: Plot of the normalized wake profiles with the different transition models and the low-resolution mesh.

Figure 58: The normalized static pressure on the vane with the three transition models and the low-resolution mesh.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 59: Plots of the normalized x-wall shear stress on the vane for the three transition models and the low-resolution mesh.

Re $2.74 \times 10^5$ – 20 Degrees

Figure 60 shows the flow separation on the pressure side and suction side of the vane for both meshes. The Gamma-Theta model shows separation on the pressure side of the vane, while the other two models do not. Figure 61–Figure 63 shows plots of the 2D wake profiles, the static pressure load curves and the x-wall shear stress for the low-resolution mesh.

Figure 60: Iso-surfaces of negative axial velocity on the suction side (SS) and pressure side (PS) of the vane. Grey areas indicate regions of separated flow. High-resolution mesh (left) and low-resolution mesh (right).

Figure 61: Plot of the normalized wake profiles with the different transition models and the low-resolution mesh.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 62: Normalized static pressure on the vane with the three transition models for flow angle 20 degrees and the low-resolution mesh.

Figure 63: Comparison of the x-wall shear stress on the vane with the different transition models for flow angle 20 degrees and the low-resolution mesh.

Re $2.74 \times 10^5$ – 25 Degrees

Figure 64 shows the flow separation on the pressure side and suction side of the vane for both meshes. The Gamma-Theta model shows separation on the pressure side of the vane with the low-resolution mesh, while the other two models do not. Figure 65–Figure 67 shows plots of the 2D wake profiles, the static pressure load curves and the x-wall shear stress for the low-resolution mesh.

Figure 64: Iso-surfaces of negative axial velocity on the suction side (SS) and pressure side (PS) of the vane. Grey areas indicate regions of separated flow. Inlet flow angle is 25 degrees. High-resolution mesh (left) and low-resolution mesh (right).
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 65: Plot of the normalized wake profiles with the three transition models for flow angle 25 degrees and the low-resolution mesh.

Figure 66: Normalized static pressure on the vane with the three transition models for flow angle 25 degrees and the low-resolution mesh.

Figure 67: Comparison of the x-wall shear stress on the vane with the three transition models for flow angle 25 degrees and the low-resolution mesh.

Re $2.74 \cdot 10^5$ – 30 Degrees

Figure 68 shows the flow separation on the pressure side and suction side of the vane for both meshes. Figure 69-Figure 71 shows plots of the 2D wake profiles, the static pressure load curves and the x-wall shear stress for the low-resolution mesh.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 68: Iso-surface of negative axial velocity indicating separation on the suction side (SS) and pressure side (PS) of the vane, for an inlet flow angle of 30 degrees with high-resolution mesh (left) and low-resolution mesh (right).

Figure 69: Plot of the normalized wake profiles with the three transition models for flow angle 30 degrees and low-resolution mesh.

Figure 70: Normalized static pressure on the vane with the three transition models for flow angle 30 degrees and the low-resolution mesh.
Comparison With rk-ε Model

Figure 72 shows an overview of the 2D wake profiles, wall shear stresses and load curves for both Fluent Transition model and the rk-ε model with all four flow angles plotted together. It can be seen that the wake profiles grow with increasing angle for both models and also that the transition point moves upstream with increasing flow angle for the transition model. As would be expected, the rk-ε model computes the boundary layer as fully turbulent and shows no transition (in the plot of the x-wall shear stress.)
Figure 72: Comparison of the rk-ε model (left) with the Fluent Transition model (right). Note that the vertical axis in the wake profile-plot starts at 0.1 for the rk-ε model and at 0.2 for the transition model.
Re $2.20 \cdot 10^5$ – Flow Quality Overview

Figure 73 shows an overview of the flow separation on the pressure side of the vane, for Re $2.20 \cdot 10^5$ and all flow angles, with the high-resolution mesh. Figure 74 shows the streamlines on the pressure side of the vane in the same way.

---

Re $3.20 \cdot 10^5$ – Flow Quality Overview

Figure 75 shows an overview of the flow separation on the pressure side of the vane, for Re $3.20 \cdot 10^5$ and all flow angles, with the high-resolution mesh. Figure 76 shows the streamlines on the pressure side of the vane in the same way.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 75: Iso-surfaces of negative axial velocity for all three transition models and all four flow angles (i.e. 10, 20, 25, 30 degrees). Grey areas show regions of separated flow. View from pressure side. Flow direction is left to right.

Figure 76: Overview of streamlines on the pressure side of the vane for all flow angles with the high-resolution mesh. Flow direction is left to right.

Re $4.80 \times 10^5$ – Flow Quality Overview

Figure 77 shows an overview of the flow separation on the pressure side of the vane, for Re $4.80 \times 10^5$ and all flow angles, with the high-resolution mesh. Figure 78 shows the streamlines on the pressure side of the vane in the same way.
Investigation of Aerodynamic Performance Predictions by CFD Using Transition Models and Comparison with Test Data

May 11, 2016

Figure 77: Iso-surfaces of negative axial velocity for all three transition models and all flow angles (i.e. 10, 20, 25, 30) with the high-resolution mesh. Grey areas show regions of flow separation. View from pressure side of the vane. Flow direction is left to right.

Figure 78: Overview of streamlines on the pressure side of the vane for all inlet flow angles with the high-resolution mesh. Flow direction is left to right.
Loss Predictions – Low-Resolution mesh

Table 7 shows the calculated 3D pressure losses for the low-resolution mesh. The test data has been removed. Table 8 shows the calculated 2D pressure losses for the low-resolution mesh in the same way. Figure 79 shows a comparison of the calculated 2D and 3D pressure losses between the high- and low-resolution mesh. The flow angle is constant at 20 degrees while the Reynolds number varies in the plots.

Table 7: 3D-losses ($dP/t\cdot q,\text{in}$) for all transition models, CFD codes, Reynolds numbers and flow angles with the low-resolution mesh. The test data has been removed.

<table>
<thead>
<tr>
<th>Re</th>
<th>Angle</th>
<th>Fluent</th>
<th>CFX-GT</th>
<th>CFX-G</th>
</tr>
</thead>
<tbody>
<tr>
<td>$2.20\cdot10^5$</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>5.51 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$2.74\cdot10^5$</td>
<td>10</td>
<td>3.67 %</td>
<td>3.82 %</td>
<td>3.72 %</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>4.67 %</td>
<td>4.15 %</td>
<td>6.40 %</td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>5.94 %</td>
<td>4.10 %</td>
<td>4.85 %</td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>7.25 %</td>
<td>7.69 %</td>
<td>6.66 %</td>
</tr>
<tr>
<td>$3.20\cdot10^5$</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>4.43 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>5.61 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$4.80\cdot10^5$</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>3.69 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 8: Integrated 2D-losses for all transition models, CFD codes, Reynolds numbers and flow angles with the low-resolution mesh. The test data has been removed.

<table>
<thead>
<tr>
<th>Re</th>
<th>Angle</th>
<th>Fluent</th>
<th>CFX-GT</th>
<th>CFX-G</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.20·10⁵</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>2.19 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.74·10⁵</td>
<td>10</td>
<td>1.82 %</td>
<td>1.97 %</td>
<td>1.93 %</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>1.85 %</td>
<td>1.97 %</td>
<td>1.68 %</td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>2.16 %</td>
<td>2.26 %</td>
<td>2.65 %</td>
</tr>
<tr>
<td></td>
<td>30</td>
<td>3.48 %</td>
<td>3.55 %</td>
<td>3.07 %</td>
</tr>
<tr>
<td>3.20·10⁵</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>1.72 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>1.94 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.80·10⁵</td>
<td>10</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>1.51 %</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 79: Comparison of calculated 3D losses (left) and 2D losses (right) between high- and low-resolution mesh for flow angle 20 and all Re.