CFD Methods for Predicting Aircraft Scaling Effects

KARL PETTERSSON

Doctoral Thesis
Stockholm, Sweden 2008
Abstract

This thesis deals with the problems of scaling aerodynamic data from wind tunnel to free flight conditions. The main challenges when this scaling should be performed is how the model support, wall interference and the potentially lower Reynolds number in the wind tunnel should be corrected.

Computational Fluid Dynamics (CFD) simulations have been performed on a modern transonic transport aircraft in order to reveal Reynolds number effects and how these should be scaled accurately. A methodology for scaling drag and identifying scaling effects in general is presented. This investigation also examines how the European Transonic Wind tunnel twin sting model support influences the flow over the aircraft.

When the Reynolds number is differing between the wind tunnel and free flight conditions, a change in boundary layer transition position can occur. In order to estimate first order boundary layer transition effects a correlation based transition prediction method, previously presented by Menter and Langtry, is implemented in the CFD solver Edge. The transition model is further developed and a novel set of equations for the production terms is found through a CFD/optimizer coupling. The transition data, used to calibrate the CFD transition model, have been extracted from a low Mach number wind tunnel campaign. At these low Mach numbers many compressible CFD solvers suffer of poor convergence rates and a deficiency in robustness and accuracy might appear.

The low Mach number effects are investigated, and an effort to prevent these is done by implementing different preconditioning techniques in the compressible CFD solver Edge. The preconditioners are mainly based on the general Turkel preconditioner, but a novel formulation is also presented in order to make the numerical technique less problem dependent.
Preface

This work was carried out at the Department of Aeronautical and Vehicle Engineering. The financial support to this work was provided by the REMFI project (contract AST3-CT-2004-502895), 6th EU framework program and is gratefully acknowledged. SNIC and PDC are also acknowledged for their help and support with the parallel computer on which these calculations have been done.

The author is very grateful to Prof. Arthur Rizzi for the support and encouragement and for giving me the opportunity to pursue a phd at the department.

My fellow co-workers at the department and my fellow graduate student in the research group Adrien and the already graduated Simone and Tomas are to be especially thanked. A special dedication goes to my former colleague Hanyo, who passed away during my time at the department. You will be missed.

Lastly and most importantly, I would like to thank my family and my bride to be Helena for always being supportive and caring. Thank you!

Dissertation

The thesis consists of an introduction and the following appended papers.


iv


Division of Work Between Authors

Paper A-C and E. Pettersson performed the computations, wrote and presented the paper. Rizzi supervised the work and contributed with valuable comments for the analysis of the results.

Paper D. Pettersson finalized the implementation of the model, wrote the optimization program, determined boundary conditions and performed the computations. Pettersson analyzed the results and wrote and presented the paper. Crippa performed the initial implementation of the transition model in the CFD code Edge.
# Contents

1 Overview and Summary

1 Introduction

1.1 Summary of Appended Papers

2 Scaling Aerodynamic Data to Free Flight Conditions

2.1 Examples of Scaling Effects

2.2 Proposed Scaling Methodology

3 Boundary Layer Transition Modeling

3.1 Correlation Based Transition Prediction Model

3.2 Results of the Transition Prediction Model

4 Compressible CFD computations at low Mach numbers

4.1 A Novel Formulation for Limiting the Precondition Parameter $\beta$

4.2 Results of CFD computations using Preconditioning

5 Conclusions

Bibliography
Part I

Overview and Summary
Chapter 1

Introduction

This thesis consists of an overview and summary section and appended papers. The first section serves as an introduction to the field of science investigated and the appended papers are previously presented material in article form. These articles have been reformatted to comply with the layout of this thesis. The overview and summary section presents some of the problems encountered. Possible solutions to these problems are presented as well. The results are discussed and critically analyzed in both the introduction and the appended papers.

1.1 Summary of Appended Papers

Some of the methods available to engineers when aircrafts are aerodynamically designed are: wind tunnels, Computational Fluid Mechanics (CFD) and semi-empirical correlations. These methods typically aims at determining the aerodynamic forces and moments of the vehicle. To predict these forces and moments early on in the design process are important in order to avoid a costly step backwards to a previous design stage. All of the methods available have different prerequisites compared to flight testing. The model tested in the wind tunnel might be smaller in size and might lack some of the geometrical details present on the full size aircraft. The wind tunnel results must also be corrected to account for the wind tunnel walls and support system, which are not present at free flight conditions. The CFD methods might also lack some of the geometrical details present on the full size aircraft. Other uncertainties of the CFD methods might be the choice of turbulence model. The semi-empirical correlations might be dependent on previously designed aircraft and might lack the information needed to develop a new type of aircraft. It is not until the final design has been built and tested (using flight tests) that it is possible to draw conclusions about which of the methods used in the design phase that was more accurate. To use all of the methods together in order to increase the confidence of the predicted aerodynamical performance of the free flying aircraft is important. Some possibilities of how this could be done is presented in
the appended Paper A. Here are examples of how CFD and wind tunnel experiments could be used together to more effectively identify scaling effects and predict free flight aerodynamic characteristics. Paper B investigates Reynolds number effects on an A380 type of aircraft. The Reynolds number effects are identified using CFD methods and a novel scaling methodology is presented. The Reynolds number based on momentum thickness \( (Re_\theta) \) is proposed to be used as identifier for possible Reynolds number scaling effects. The applicability to use CFD methods to predict viscous boundary layer effects are presented in paper C. One of the challenging tasks when aerodynamic data should be scaled to higher Reynolds number is how the boundary layer transition position changes. One method to estimate first order transition effects are presented in Paper D. The transition prediction method is calibrated to low Mach number wind tunnel results. It was shown that at these low Mach number flows the compressible CFD solver Edge experienced some problems with robustness. Paper E presents developments of a preconditioning technique, to increase robustness and accuracy for low Mach number flows.
Chapter 2

Scaling Aerodynamic Data to Free Flight Conditions

Today wind tunnel testing and CFD calculations are natural and necessary parts in the development of an aircraft. The trends are that more and more time is spent in the wind tunnel and performing CFD calculations [1]. This is performed in order to avoid a costly step backwards to a previous design phase to correct for potential mistakes. The main goal of the CFD calculations and the wind tunnel testing is to predict the free flight aerodynamics of the aircraft. Accurate flight performance prediction is a challenging task because most of the testing has been done at sub-scale conditions. Some of the phenomena which have to be taken into account when wind tunnel data is scaled to free flight conditions are: the influence of the wind tunnel wall and model support interference effects and potentially a lower Reynolds number in the wind tunnel. Aeroelastic effects might also differ between the wind tunnel model and the aircraft. Some recent research of scaling methodology is given by Eckert [2], where the influence of wall and sting interference and the impact of the propeller of the Airbus A-400M is investigated. The high cost associated with acquiring free flight test data makes the amount of information about scaling ground to flight methodology rare in the open literature and are often company proprietary and part of their competitive edge [3]. Some of the drivers of an increased accuracy in scaling methodology are the economical benefit from having an optimal choice of engine for a given aircraft configuration and the increased need of reduction of emissions [4]. An erroneously predicted scale effect which would imply an increase in drag of 1% for an ultra high capacity aircraft would equate to around 3 tonnes of extra fuel at constant range or a reduction in range of 120 km for constant maximum take off weight [5].

The scaling effects can introduce an element of risk in the aircraft program, particularly for large wings, which are designed for high subsonic Mach numbers. An investigation of the scale and Reynolds number effects could increase the development costs drastically if they were to be done in the wind tunnel only [4].
Using CFD methods as a complement to wind tunnel testing can therefore be economically beneficial. Modern wind tunnel testing techniques and CFD methods also complement each other with the high fidelity of the wind tunnel results and the extensive data set (capable of free flight Reynolds and Mach number) from the CFD calculations which enables a thorough investigation of flow topology and phenomena.

Historically the importance of matching the free flight Mach number has never been in question [5]. Today it is also possible to match free flight Reynolds number in cryogenic facilities like the National Transonic Facility (NTF) [6] in USA and the European Transonic Wind tunnel (ETW) [7] in Europe. CFD methods and cryogenic wind tunnel testing are typically being incorporated early in the process when designing a modern transonic transport. The currently developed Boeing 787 was tested in the ETW early on in the design process and the final determination of how well the wind tunnel data matches flight performance will follow from flight testing [8].

CFD methods and wind tunnel measurements both have their own uncertainties. Using viscous CFD methods to predict high Reynolds number flows could for instance be dependent on which turbulence model that is used. Since no, computationally viable, universal turbulence model exists, the best model for the problem at hand needs to be chosen. Even before the CFD calculations have started uncertainties are introduced due to the turbulence model or boundary conditions chosen or the level of detail in the geometrical representation. The wind tunnel corrections are in question as well and should according to Rasuo [1] not be taken as the “exact value”. The success in estimating scaling and Reynolds number effects has however increased in later years due to the extensive use of cryogenic wind tunnels together with the use of modern CFD methods [9].

A recent example of this is given by Nicoli [10] where the VEGA launcher (covering a wide range of Reynolds and Mach number) was scaled to free flight conditions using several different wind tunnels and modern CFD methods. A systematic error between CFD and wind tunnel results was identified and corrections could be done to scale the aerodynamic data.

### 2.1 Examples of Scaling Effects

The well-known example of a Reynolds number effect which drastically made the transonic performance worse is the C-141. A plot of the super-critical pressure distribution and a schematic view of the boundary layer is shown in figure 2.1. The wind tunnel transition was fixed in the front of the airfoil and the agreement in transition position (in percent of chord-wise position) to the free flight condition was good. The larger Reynolds number at free flight conditions however made the relative thickness of the boundary layer smaller since the thickness of a turbulent boundary layer typically scales with $Re^{-1/5}$. This effect is shown in the schematic view in figure 2.1. The thinning of the boundary layer moves the shock in stream-
wise direction, closer to the trailing edge. The region of shock induced separation decreases in size. If $C_L$ is kept constant for a given Mach number with increasing Reynolds number, the increased aft loading must be compensated with a decrease in the load over the front of the airfoil. This is generally accomplished by a decrease in the angle of attack [12].

Saltzman [13] summarized in his work from 1982, some of the known scaling issues at the time. The results for different aircrafts and discrepancies between wind tunnel and flight conditions are shown in table 2.1. Several other scaling phenomena are discussed in ref. [13], [14], [15] and [11]. All of the discrepancies in table 2.1 occur at transonic speeds and are influenced by one or more of the following effects:

- wall interference effects
- disproportionate boundary layer (Reynolds number effects)
- sting-support interference effects.

In the work of Lessard [16] the high-lift Technology Concept Airplane (TCA), which were part of the High Speed Research program, were investigated using wind tunnel measurements and CFD calculations. Here is the aircraft alone, with wind tunnel support system and with support system and wind tunnel walls modelled and calculated using CFD methods. Comparing global forces and moments shows that the CFD calculations with the aircraft alone produces results which are in closer agreement to the corrected wind tunnel results. This is especially evident when pitching moment are being compared. Computations like these could give additional information on how newly developed wind tunnel support systems affects the flow and on scaling effects in general.

Another example where wind tunnel mounting effects are investigated comes from the work done by Pettersson et al. [17]. This work was carried out within
Aircraft Discrepancy Apparent cause Remarks

P-51 Flight drag after Different Believe related pullout higher separation to discussion than for model locations of C-141 and M-2/F-3

X-5 Drag difference Chubby body, Probably differing at Ma 1, though different afterbody flow the same at drag separation divergence Ma locations

M-2/F-3 Base drag and Chubby body, Compensating boattail drag effects; fortuitous different separation locations

X-15 Base drag Sting-affected Eliminated variable base pressure by subtracting out

XB-70 Model drag too Tunnel wall Flexibility effects low at Ma 1.18 may also have effects contributed

F-8 Second-velocity Tunnel wall Model too large, peak larger and too close farther aft in flight to Ma 1

Table 2.1: Summary of wind tunnel model/flight discrepancies (Saltzman [13]).

the REMFI [18] project. An A380 type of aircraft was investigated for increased knowledge in empennage design and flow physics. Tail stall characteristics, elevator and rudder efficiency, scale effects, gap effects and aeroelastic effects, amongst other empennage design characteristics, were investigated using a “live rear-end” measuring technique. In the work done by Pettersson et al. [17] three different Reynolds number were evaluated, 20, 38 and 56 million. The free stream Mach number was held constant at 0.85 with a constant angle of attack of 0°. CFD computations were performed with and without the ETW twin sting booms mounted. One way of examining the flow topology is shown in figure [2.2]. Here, the wing is seen from above, colored by $c_p$ and iso-lines of $c_p$. The two cases (with and without booms) both have the same range of $c_p$ in order to make it easier to localize pressure changes. At half span of the wing it is clearly visible that the flow is influenced by the presence of the boom. The structure of the flow at the root and tip seems however to be intact. A large low pressure region is observed close to where the boom is mounted. By analyzing how the flow is disturbed by the twin sting boom, additional conclusions and credibility to the empennage scaling phenomena identified in the wind tunnel campaign can be achieved.
2.2 Proposed Scaling Methodology

The method of extrapolating drag presented here is taken from the work done within the REMFI project. Details of this scaling procedure could be found in Pettersson et al. [19], the outline of the method is presented below.

Assume that equation (2.1) is valid.

\[
\left( \frac{C_{D_p}}{C_F} \right)_{WT} = \left( \frac{C_{D_p}}{C_F} \right)_{CFD}
\]

This means that the ratio of drag due to skin friction and drag due to pressure are the same when comparing wind tunnel and CFD results. This does not however mean that absolute numbers in drag have to match. If the left hand side of equation (2.1) is known thru CFD calculations the drag due to pressure and skin friction from the wind tunnel results could be evaluated \((C_{D,wt} = C_{D_p,wt} + C_{F,wt}\) and the ratio of the two on the right hand side is known and \(C_{D,wt}\) known from the wind tunnel test). When the two components of drag are known (drag due to normal and drag due to tangential forces) they could be scaled separately with Reynolds number. As shown in Pettersson et al. [19] the \(C_{D_p}\) part of drag has a Reynolds number dependence as well and this scaling should not be lumped into the complete scaling of drag using a semi-empirical relation for skin friction. One of the Reynolds number effects on \(C_{D_p}\) for the transonic transport investigated in [19] was that the region of separated flow in the fuselage/horizontal tail plane junction decreased with increasing Reynolds number and this in turn affected the drag due to pressure. The drag due to skin friction was scaled using the semi-empirical Karman-Shoenherr correlation, see Covert [20] for example, together with the Sommer-Short method [21] to account for compressible effects. Drag due to
pressure was extrapolated using equation (2.2).

\[ C_D = C_1 + C_2(Re)^n \] (2.2)

The three coefficients in equation (2.2) were found by investigating three different Reynolds numbers and performing a least-square fit to the data. Figure 2.3 show total drag, drag due pressure and skin friction of the empennage. Forces and moments were measured using a “live rear-end technique” isolating the empennage alone. The data has been anchored to the 38 million wind tunnel data case. The extension of the total drag curve would correspond to scaling drag to free flight Reynolds number. In figure 2.3 are the ARA and ETW wind tunnel data shown.

Figure 2.3: Extrapolation of \( C_D \) to free flight Reynolds number conditions, all data anchored to the ETW 38 million data point (left: total drag, middle: drag due to pressure and right: skin friction).

The ARA data are available at 3 million Reynolds number and the ETW data are available at 3, 20 and 38 million Reynolds number. At the low Reynolds number case the transition were fixed at 10% chord of the htp using a transition strip while at the higher Reynolds number cases no trip strip were applied and free transition occurred.

In order to compare the scaling method to the highest available Reynolds number wind tunnel data, the drag extrapolation procedure were done as shown above with the exception that it was anchored to the 20 million Reynolds number wind tunnel case instead. This gave the opportunity to compare the extrapolated data to actual wind tunnel data at 38 million Reynolds number, the extrapolated data and the wind tunnel result differed with approximately one drag count. Using separate scaling procedures for skin friction and drag due to pressure were assumed to be more accurate for higher angle of attack cases.
Chapter 3

Boundary Layer Transition Modeling

An erroneously estimated boundary layer transition position could give large consequences on the scaling of the aerodynamic data acquired at wind tunnel conditions. In the C-141 example in chapter 2 the erroneously estimated viscous boundary layer resulted in a large shift in chord-wise position of the shock. Some examples in aircraft design where an accurate prediction of the transition position is important are low Reynolds number airfoils and hypersonic reentry spacecraft. The low Reynolds number airfoils could be mini- or high altitude Unmanned Aerial Vehicles (UAV’s). Here a large portion of the airfoil could be laminar. The performance of the aircraft could seriously be mistaken if the laminar part of the wing is badly estimated. In the spacecraft example, the importance of predicting the transition position is due to the fact that laminar/turbulent boundary layers have different heating rates, which seriously affects the construction of the heating shields.

Boundary layer transition prediction is also an important topic when constructing turbines used for aircraft engines. Here are two main areas of interest [22]:

- Effect of temperature distribution
- Varying Reynolds number comparing takeoff and cruise conditions

The effect of varying temperature distributions, when comparing laminar and turbulent boundary layers, significantly affects the life prediction of blades and vanes in high pressure turbines. The blade temperature is also an important design criterion which affects the performance of the engine. The second point is due to the reduction in performance degradation when comparing the takeoff to the cruise conditions of the engine. At sea-level takeoff conditions the Reynolds number is lower than compared to the high altitude cruise conditions of the engine. It is important to understand these effects when designing a fuel and weight efficient aircraft engine.
In order to estimate the transition position, methods like DNS/LES, low Reynolds number two equation models, $e^N$, or semi-empirical/wind tunnel correlations are available to the engineer. The use of DNS or LES are too computationally expensive to use for either aircraft or engine design, when comparing to the capacity of computers today. The low Reynolds number RANS modeling concept uses wall damping functions to trigger the transition onset.

This approach is however not capable of reliably capturing the influence of the many factors influencing transition, where some are:

- Free stream turbulence
- Pressure gradients and separation
- Mach number
- Turbulent length scale
- Wall roughness
- Streamline curvature

It is however not surprising that the low Reynolds number model is not able to capture these effects since the damping functions have been optimized to damp the turbulence in the viscous sublayer and not to capture a completely different physical process, such as boundary layer transition [23].

The $e^N$ method has the drawback that it can not predict non-linear transition phenomena such as bypass transition [24]. It is also an open question how the $N$ factor should be chosen for any given problem.

The semi-empirical methods are based on approximations to wind tunnel data such as the Abu-Ghannam and Shaw [25] correlation. Using these methods in general CFD methods could be hard since integrated boundary layer information could be difficult to calculate in general CFD cases where a clear definition of the boundary layer edge might be missing.

### 3.1 Correlation Based Transition Prediction Model

Menter et al. [23] developed a correlation based transition prediction model, dependent on local flow variables only, to predict first order transition effects for some of the phenomena mentioned previously. The method was developed to be easily incorporated into existing CFD codes, and the requirements for the transition prediction model were determined to be [26]:

- Allow the calibrated prediction of the onset and the length of transition
- Allow the inclusion of different transition mechanisms
- Be formulated locally (no search or line-integration operations)
• Avoid multiple solutions (same solution for initially laminar or turbulent boundary layer)
• Do not affect the underlying turbulence model in fully turbulent regimes
• Allow a robust integration down to the wall with similar convergence as the underlying turbulence model
• Be formulated independent of the coordinate system
• Applicable to three-dimensional boundary layers.

This transition prediction model has also been developed and tested in the work by: Misaka et al. [27], Gürdamar et al. [28], Sørensen [29] and Pettersson et al. [30].

An overview of the correlation based transition prediction model by Menter et al. [23] is presented below. The transition model utilizes two extra transport equations, in addition to the $k-\omega$ equations, see equations (3.1) and (3.2). To model non-local effects, a transition onset criteria $(Re\theta_t)$, which depends on local flow properties such as acceleration parameter and turbulence intensity, is convected in the free-stream and diffused into the boundary layer. $Re\theta_t$ is then coupled to the local intermittency ($\gamma$) which is modelled with the $\gamma$ transport equation. Production of $\gamma$ is promoted when the local vorticity number exceeds a critical $Re\theta_t$ ($Re_{\theta_c}$). The intermittency is finally coupled to the production of $k$ in the $k-\omega$ SST model in order to form a closed system of equations. Other turbulence models than the $k-\omega$ SST model could be used. Lodefier et al. [31] describes the differences between the $k-\epsilon$ and the $k-\omega$ models in general, and their applicability to be used together with extra transport equations to model boundary layer transition. Lodefier et al. [31] favors the $k-\omega$ model due to the explosive nature of the production terms in the $k-\epsilon$ model.

$$\frac{\partial (\rho \gamma)}{\partial t} + \frac{\partial (\rho U_j \gamma)}{\partial x_j} = P_\gamma - E_\gamma + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial (\gamma)}{\partial x_j} \right]$$

(3.1)

$$\frac{\partial (\rho Re\theta_t)}{\partial t} + \frac{\partial (\rho U_j Re\theta_t)}{\partial x_j} = P_{\theta_t} + \frac{\partial}{\partial x_j} \left[ \sigma_{\theta_t} (\mu + \mu_t) \frac{\partial (Re\theta_t)}{\partial x_j} \right]$$

(3.2)

The production term $P_\gamma$ for the intermittency ($\gamma$) equation is calculated as,

$$P_\gamma = F_{\text{length}} c_{a_1} \rho S (\gamma F_{\text{onset}})^{0.5} (1 - c_{e_1} \gamma)$$

(3.3)

where

$$F_{\text{onset}} = \max(F_{\text{onset},2} - F_{\text{onset},3}, 0) \quad F_{\text{onset},1} = \frac{Re_V}{2.193 \cdot Re_{\theta_c}}$$

$$F_{\text{onset},2} = \min(\max(F_{\text{onset},1}, F_{\text{onset},4}^4), 2.0)$$

$$F_{\text{onset},3} = \max(1 - \left( \frac{Re_T}{2.5} \right)^3, 0)$$

(3.4)
The vorticity Reynolds number \((Re_V)\), based on strain rate, and the turbulent viscosity ratio \((R_T)\) is calculated as

\[
Re_V = \frac{\rho y^2 S}{\mu} \quad R_T = \frac{\rho k}{\mu \omega}
\]  

(3.5)

where \(y\) is the wall distance and \(S\) the strain rate magnitude. The local formulation is based on the fact that a scaled Blasius boundary layer follows:

\[
\frac{\max(Re_V)}{2.193 \cdot Re_{\theta_i}} = 1
\]  

(3.6)

If \(Re_V\) in the boundary layer exceeds \(2.193 \cdot Re_{\theta_i}\) the \(F_{onset,1}\) term becomes active and transition is promoted. The \(Re_{\theta_i}\) term is a critical Reynolds number based on momentum thickness, and could be thought of as the position where turbulence starts to grow. \(Re_{\theta_i}\) could be thought of where the turbulent boundary layer starts to transition from a laminar to a turbulent profile. The \(F_{onset,3}\) term is added to ensure that the transition model is active through the entire transitional region.

For pressure gradient flows equation (3.6) no longer holds. For moderate pressure gradients however the difference between the actual momentum thickness and the vorticity Reynolds number is reported (see ref. [26]) to be less than 10%.

The destruction term \(E_{\gamma}\) is determined by,

\[
E_{\gamma} = c_{a_2} \rho \Omega F_{turb} (c_{e_2} \gamma - 1)
\]  

(3.7)

where

\[
F_{turb} = e^\left(\frac{R_T}{4}\right)^4
\]

This makes the destruction of \(\gamma\) active inside boundary layers only. The boundary conditions of \(\gamma\) are one at free stream boundaries and zero flux at walls. When the intermittency is one, the underlying turbulence model remains unchanged, and locally generated turbulence is allowed to be convected downstream. Using \(\gamma\) equal to one in the free stream allows, in the example of a wing with a flap, that the turbulence generated over the wing will promote transition earlier on the flap. For further details of the \(\gamma\) distribution in the boundary layer/free stream region see figure 3.2.

The production term \(P_{\theta_i}\) for the transition onset criteria \(Re_{\theta_i}\) is calculated as,

\[
P_{\theta_i} = c_{\theta_i} \frac{\rho}{t} (Re_{\theta_i} - Re_{\theta_i}) \cdot (1 - F_{\theta_i})
\]  

(3.8)

where

\[
F_{\theta_i} = \min \left( \max \left( F_{wake} \cdot e^{-\left(\frac{y}{\delta}\right)^4}, 1 - \left( \frac{\gamma - 1/c_{e_2}}{1 - 1/c_{e_2}} \right)^2 \right), 1 \right)
\]
and

\[ t = \frac{500\mu}{\rho U^2} \quad F_{wake} = e^{-\left(\frac{u_{wake}}{100}\right)^2} \quad Re_\omega = \frac{\rho \omega y^2}{\mu} \]

\( \delta \) is finally calculated as

\[ \delta = 50\Omega y \frac{\delta_{BL}}{U} \quad \delta_{BL} = \frac{15}{2} \theta_{BL} \quad \theta_{BL} = \frac{Re_\theta \mu}{\rho U} \]  

(3.9)

The \( F_{\theta_t} \) term is one in the boundary layer and zero in the free stream, allowing production in the free stream only. \( Re_\theta \) is then diffused into the boundary layer where the rate of diffusion is determined by \( \sigma_\theta \). This creates a lag between the free stream value of \( Re_\theta \) compared to inside the boundary layer. A lag between the free stream value compared to inside the boundary layer is desirable because the onset of transition is primarily affected by history effects and not by local flow variables. How large this lag should be is determined through numerical experiments varying \( \sigma_\theta \) and comparing the results to wind tunnel data [26]. The production term \( P_{\theta_t} \) drives \( Re_\theta \) towards a semi-empirical correlation for \( Re_\theta \). \( Re_\theta \) depends on local flow parameters such as turbulence intensity and acceleration of the flow, one example of a \( Re_\theta \) correlation is the one produced by Abu-Ghannam and Shaw [25]. Different correlations have been tested, including the original formulation presented by Menter et al. [23] and the most recently developed by Langtry [26]. The results in figure 3.1 is produced by using the original formulation of \( Re_\theta \) as presented in ref. [23].

The two terms, \( F_{length} \) and \( Re_\theta \), are not described in [23] or [26], due to proprietary reasons. One way to determine these functions is presented in Pettersson et al. [30]. Here the \( F_{length} \) and \( Re_\theta \) terms are calibrated to the Ercoftac T3A, T3A- and T3B wind tunnel results [32].

Figure 3.1 shows the results from Pettersson et al. [30] when \( F_{length} \) and \( Re_\theta \) have been optimized to minimize the difference between \( c_f \) from CFD and wind tunnel experiments. The model criterion (allow the transition model to be calibrated to experiments) is shown to be fulfilled for all cases except the T3B case. All efforts to match the correlation based transition prediction model to the T3B case seem to give similar results whether constant values for \( F_{length} \) and \( Re_\theta \) or functional expressions are used, see ref. [29]. The results in figure 3.1 are similar to the results from other researchers in ref. [23] [27] [28] [29].

### 3.2 Results of the Transition Prediction Model

The results from the calibration of the transition prediction model are shown in figure 3.1. Transition, local skin friction, is analyzed for the flat plate, zero pressure gradient flow. As free stream turbulence and Reynolds number is varied, the transition region is shifted along the flat plate. The free stream Reynolds number per
Figure 3.1: Comparison of $c_f$ between wind tunnel and CFD calculations for the T3 test cases (Pettersson et al. [30]).

Meter, in figure 3.1, ranges from $3.6 \cdot 10^5$ to $1.3 \cdot 10^6$ and the turbulence intensities, measured at the flat plate leading edge ($T_{ul.e}$), ranges from 0.93% to 6.1%.

The development of intermittency over the flat plate T3A case is shown in figure 3.2. Here it is clear that the intermittency is low inside the laminar boundary layer and one in the free stream. The flat plate has a round leading edge with radius 0.75 mm, as a reference to other dimensions in the figure. The lower value of intermittency, 0.02, is determined by the constant $c_e$ in equation 3.7.

Figure 3.2: Intermittency ($\gamma$) at the flat plate leading edge, T3A conditions.

$Re_{\theta}$, from the same calculation as figure 3.2, is presented in figure 3.3. Here an increase in $Re_{\theta}$ is seen as the flow progresses downstream (flow from left to right). This is due to the decay of turbulence intensity downstream of the inlet boundary condition, which increases the local transition onset criteria.

The transition model does not attempt to model the physics of the transition
process (unlike e.g. turbulence models), but form a framework for the implementation of transition correlations into general purpose CFD methods [26]. One of the requirements set in the original description of the transition model is: to have a local formulation where no integrated quantities should be calculated (such as streamline or momentum thickness integration). This makes the model suitable for structured and unstructured solvers on modern CFD codes with massive parallel execution. One attempt to extend the model to also incorporate compressible effects is presented by Kaynak et al. [33]. Here, the $Re_{\theta_t}$ term is multiplied with a function depending on the local Mach number. This results in a delay of transition, for increasing free stream Mach number. Mach numbers up to 1.2 are investigated in the report. An alternative to this approach is to incorporate the compressible correction in the production term of $\gamma$ as in the work of Lodefier et al. [31]. This work aims at modelling bypass transition and uses only one extra transport equation ($\gamma$).

Figure 3.3: Transition onset criteria ($Re_{\theta_t}$), T3A conditions.
Chapter 4

Compressible CFD computations at low Mach numbers

According to Pierce et al. [34] CFD methods have developed in response to the need for accurate, efficient and robust numerical algorithms for solving increasingly complete descriptions of fluid motion over increasingly complex flow geometries.

One of these requirements, which cannot be compromised, is accuracy. If accuracy is lost and a more or less random solution is achieved, then also the possibility to draw conclusions about trends in series of CFD calculations is lost. The CFD calculations might not give the “right answer” in an absolute sense, but should be assumed to provide a solution within an acceptable accuracy bound [35]. It has however been pointed out that CFD is able to compute delta drag levels between similar configurations very well and that this was how CFD generally was used in the industry for the design and development of aircrafts [36].

One of the most important aspects of any computational method is robustness [34]. The computational method should, ideally, be able to compute two similar cases using approximately the same amount of time. The user specified input parameters should, to a minimum, have to be varied between the two similar cases. The user specified input parameters should, ideally, affect one numerical algorithm or physical quantity modelled in the solution at a time.

The possibility to use one code for many different applications and flow conditions makes it easier to compare results and it does not force engineers to learn many different CFD codes.

Low Mach number flows could be of interest when there is mixed compressible and incompressible flow present, e.g. flow over an airfoil at moderate Mach numbers. The need for solving low Mach number flows could also appear when new turbulence models should be calibrated in the CFD code and all of the data are gathered at low Mach number wind tunnel conditions.

Slow convergence rates and oscillating residuals could be observed when compressible CFD solvers are used to calculate low Mach number flows [37].
In order to still be able to use the compressible CFD solvers, a robust and effective preconditioner is required. Robustness has been a problem with some of the preconditioning methods available. This is manifested by some of the precondition parameters being problem dependent [38].

4.1 A Novel Formulation for Limiting the Precondition Parameter $\beta$

The preconditioning matrix have terms which are inversely proportional to the local velocity and to avoid division by zero, the local velocity in the preconditioning matrix is limited by a fraction of the free stream velocity, $K \cdot U_\infty$. $K$ is rather problem dependent and is reported in Turkel [38] to vary from typically 0.4 in inviscid cases, 1.0 for viscous cases and as large as 3.0 for more difficult cases. In order to increase the robustness and accuracy of the solution, efforts to localize the limiting on $\beta$, have been performed by authors in ref [39], [38] and [40] amongst others.

Turkel [38] proposes a viscous correction, $\tilde{Re}_\Delta$, to the limiting of the preconditioning parameter $\beta$. This work adds the pressure correction $\tilde{c}_p$-term:

$$\beta^2 = \min((1 + \tilde{c}_p + \tilde{Re}_\Delta)\beta_{org}^2, c^2)$$

(4.1)

The $\beta_{org}^2$ term in equation (4.1) is the original formulation of $\beta$ as reported in ref [41] and $c$ is the speed of sound.

The pressure correction, $\tilde{c}_p$, is active when the pressure coefficient, $c_p$, is larger than 0.3. The viscous and pressure corrections are added to increase the precondition parameter $\beta$ in regions where necessary. Figure 4.1 shows the viscous and pressure corrections on $\beta$. These corrections increase $\beta$ in regions where viscous effects are dominant (low $Re_\Delta$) and where $c_p$ is high. In compressible regions the preconditioner is switched off.

4.2 Results of CFD computations using Preconditioning

An example is presented here to further illustrate the effects of accuracy, robustness and speed up when a preconditioner is used in the compressible CFD solver Edge [42].

The Aerospatiale-A airfoil (called A-airfoil) described in the ECARP initiative [43] is used as test case. This is a non-symmetric airfoil discretized with a block-structured mesh using the ICEM CFD software [44]. The free stream conditions used are, $3.4^\circ$ angle of attack, $Re_\infty$ of $3.13 \cdot 10^6$ and two Mach numbers: 0.15 and 0.015. Transition is fixed at $x/c = 0.3$ on the pressure side and at $x/c = 0.12$ on the suction side.

A summary of the A-airfoil computations performed are presented in table 4.1 MSES [45] and wind tunnel results (from the F1 wind tunnel campaign) are presented as reference to the Edge CFD computations. Convergence is defined as
when $c_d$ varies with less than 0.1% compared to the result at the last iteration, and the residuals last oscillation (at least one period) should be less less than 0.1% in magnitude of the result at the last iteration. A total of 50 000 iterations were performed. The cases which diverge, the solution actually blows up, are labeled div. in table 4.1.

Figure 4.2 shows the convergence history of the converged solutions from table 4.1. The major observations from the results in table 4.1 and figure 4.2 are that drag is decreased with almost one drag count (one count is $1 \cdot 10^{-4}$) when the preconditioner is used at the $M_\infty = 0.15$ condition and that the computation without precondition diverges for the lower Mach number investigated.

The region of influence of the viscous and pressure corrections on the A-airfoil are shown in figure 4.3. The pressure correction is shown only to be active in the stagnation point region, and the viscous correction is active throughout the boundary layer and wake region. To localize the lower limiting on $\beta$ is important.
Figure 4.2: Convergence history for the A-airfoil.

Figure 4.3: Viscous and pressure corrections on $\beta$.

from an accuracy and robustness point of view [38], [37].
Chapter 5

Conclusions

In this thesis aerodynamic scaling effects are investigated. Paper A presents a thorough investigation of scaling effects previously identified. A system of how to classify different scaling effects, in order to correct for each phenomena only once, is also presented. Examples are also given of how CFD and wind tunnel data could be used together in order to effectively identify scaling effects and predict free flight aerodynamic characteristics.

In Paper B a transonic transport type of aircraft is investigated. Reynolds number effects are identified using CFD methods and a novel scaling methodology to extrapolate drag to free flight conditions is presented. The Reynolds number based on momentum thickness ($Re_\theta$) is proposed to be used as identifier for possible Reynolds number scaling effects. It is also shown that it is possible to capture the approximate linear increase of $Re_\theta$ with increasing free stream Reynolds number using modern CFD methods.

Paper C shows the accuracy of two different CFD solvers and two different turbulence models in predicting momentum thickness for varying Reynolds number and pressure gradient. It is shown that using a modern CFD solver with low artificial dissipation, the Spalart-Allmaras and $k-\omega$ SST turbulence model, with a mesh constructed using industrial best practices, produces results of local viscous boundary layer characteristics with the same order of accuracy as achieved using modern wind tunnel measurement techniques.

Paper D presents how a correlation based transition prediction model is implemented in the compressible CFD solver Edge. The model is calibrated to wind tunnel results and the numerical model is optimized to minimize the difference between CFD and wind tunnel results for local skin friction in the transitional region. The calibration is done by coupling the CFD solver to a Matlab optimization routine. All computations in the optimization procedure are run in batch mode, unsupervised, and at low Mach numbers. At these conditions a robust and accurate preconditioning technique is mandatory in order to run all cases with restrictions given on CPU-time available. Possible ways to further develop this work is to couple
the transition model to more advanced turbulence models and validate it to several test cases, including low turbulence intensities and 3-D bodies with and without pressure gradients. Adding corrections for cross-flow instabilities and compressible effects would also add value and increase the applicability of the transition model.

Paper E presents a strategy to make the Edge code more robust for low Mach number flows. This is done by implementing a preconditioning method based on the pressure temperature variables with a viscous correction and a novel formulation for a pressure correction taking the stagnation problem into account.
Bibliography


