IMPLEMENTATION AND INVESTIGATION OF A LOCAL CORRELATION-BASED TRANSITION MODEL IN A PARALLEL NEWTON-KRYLOV-SCHUR ALGORITHM

by

Michael Piotrowski

A thesis submitted in conformity with the requirements for the degree of Masters of Applied Science
Graduate Department of Institute for Aerospace Engineering
University of Toronto

© Copyright 2016 by Michael Piotrowski
Abstract

Implementation and Investigation of a Local Correlation-based Transition Model in a Parallel Newton-Krylov-Schur Algorithm

Michael Piotrowski
Masters of Applied Science
Graduate Department of Institute for Aerospace Engineering
University of Toronto
2016

Aircraft with significant regions of natural laminar flow provide the opportunity for reduced fuel consumption. However, many conventional computational fluid dynamics algorithms assume that the flow is fully turbulent, which limits the development of natural-laminar-flow designs. A three-dimensional Reynolds-averaged Navier-Stokes flow solver has been expanded to include the prediction of laminar-turbulent transition. This was achieved by coupling the Spalart-Allmaras turbulence model with the $\gamma - \tilde{Re}_{\theta t}$ local empirical correlation-based transition model. The stability and robustness of the solution algorithm was optimized in order to obtain efficient and reliable convergence. The performance of the algorithm was investigated through application to several two-dimensional verification and validation cases, including flat-plate test cases, natural-laminar-flow airfoils, and a multi-element high-lift airfoil configuration. The results demonstrate the ability of the model to predict locations of transition in complex flow regimes which match experimental results for cases involving natural and separation-induced transition.
Acknowledgements

First and foremost I would like to thank Professor David Zingg for his endless support throughout the completion of my degree. His thought provoking questions challenged me to develop a deeper understanding, while his insightful suggestions drove me forward with my research. I would also like to thank Professor Craig Steeves and Professor Prasanth Nair for their support as members of my research assessment committee, and Professor Masayuki Yano for his assistance in the completion of this thesis.

I am thankful for the UTIAS community, and the members of the CFD lab, for providing an amazing research environment. Specifically I’d like to thank David Brown for his thoughtful discussions, Shariar, Tom, and Jenmy for their guidance and support, and Howard for his time and patience when introducing us to the Jetstream. Tim and Gregg thank you for all the magic and downtown adventures which have made the last two years an incredible experience. Shoutout to the shockwaves, ultimate frisbee group, and basketball crew.

Finally, I am forever grateful for my family and their endless love and support. I would not be who or where I am today without them.

Michael Piotrowski
# Contents

1 Introduction .................................................. 1
  1.1 Motivation ................................................. 1
  1.2 Boundary-Layer Transition ................................ 2
    1.2.1 Natural Transition .................................... 2
    1.2.2 Bypass Transition .................................... 2
    1.2.3 Separation-Induced Transition ....................... 3
    1.2.4 Relaminarisation .................................. 3
  1.3 Transition Prediction and Modelling Techniques ............. 3
    1.3.1 Direct Numerical Simulations and Large-Eddy Simulations 4
    1.3.2 \(e^N\) Method ......................................... 4
    1.3.3 Low-Reynolds-Number Turbulence Models ............... 5
    1.3.4 Empirical Correlations ................................ 6
  1.4 Flow Solver ................................................ 7
  1.5 Thesis Objectives and Outline ............................. 7

2 Governing Equations ........................................... 9
  2.1 Navier-Stokes Equations .................................. 9
  2.2 Spalart-Allmaras Turbulence Model ......................... 11
  2.3 \(\gamma-Re_{th}\)-Spalart-Allmaras Model ................... 13
    2.3.1 Model Formulation .................................... 13
    2.3.2 Curvilinear Coordinate Transformation ................ 20
    2.3.3 Initial Conditions ................................... 21
    2.3.4 Boundary Conditions ................................. 21

3 Spatial Discretization ......................................... 23
  3.1 Domain Decomposition .................................... 23
  3.2 Summation-by-Parts Operators ............................ 23
  3.3 Simultaneous Approximation Terms ......................... 26

4 Solution Algorithm ............................................ 29
  4.1 Newton-Krylov Approach .................................. 29
    4.1.1 Approximate-Newton Phase ............................ 30
    4.1.2 Inexact-Newton Phase ................................ 31
    4.1.3 Switching Between Phases ............................ 32
4.2 Preconditioning ......................................................... 33
4.3 Equation and Variable Scaling ........................................ 33

5 Algorithm Optimization .............................................. 37
5.1 Test Cases ............................................................ 37
5.2 Solution Algorithm Parameter Selection and Modifications .......... 38
  5.2.1 Time Step Considerations ......................................... 39
  5.2.2 Damping of the Solution Update ................................... 43
  5.2.3 Switch from Approximate to Inexact-Newton Phase ................. 46
5.3 Negative Turbulence Model Formulation ................................ 47
5.4 Computational Cost .................................................. 48

6 Results ................................................................. 51
6.1 Flat Plate Simulations ................................................ 51
6.2 Two-Dimensional Airfoils ............................................. 54
  6.2.1 NLF-0416 Airfoil .................................................. 54
  6.2.2 S809 Airfoil ....................................................... 57
6.3 Two-Dimensional Multi-Element High-Lift Configurations ............. 59
  6.3.1 NLR-7301 Two-Element Airfoil ................................... 59

7 Conclusions and Recommendations .................................. 63
7.1 Conclusions .......................................................... 63
7.2 Recommendations .................................................... 64

References .............................................................. 67
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>Grids and geometries for test cases</td>
<td>37</td>
</tr>
<tr>
<td>5.2</td>
<td>Flow conditions for test cases</td>
<td>38</td>
</tr>
<tr>
<td>5.3</td>
<td>Convergence performance parameters for Test Case 2A with and without transition prediction, ( \Delta t_{\text{max},dt} ) and ( R_{\text{max},dt} ) values of 10 and 1, respectively, variable clipping, an adaptive reference time step, and an ( R_d ) value of ( 1 \times 10^{-5} ). Reference time step values of ( a = 0.001 ) and ( b = 1.05 ) were used for both cases.</td>
<td>50</td>
</tr>
<tr>
<td>6.1</td>
<td>Experimental location of transition onset for zero-pressure-gradient flat plate [46]</td>
<td>52</td>
</tr>
<tr>
<td>6.2</td>
<td>Grids and geometries for natural-laminar-flow airfoils</td>
<td>54</td>
</tr>
<tr>
<td>6.3</td>
<td>Experimental and computational locations of transition onset for NLF0416 airfoil at positive angles of attack [50]</td>
<td>55</td>
</tr>
<tr>
<td>6.4</td>
<td>Location of transition onset for NLF0416 airfoil at negative angles of attack calculated by Schucker [47] and in the current work</td>
<td>57</td>
</tr>
<tr>
<td>6.5</td>
<td>Computed and experimental locations of transition onset for S809 airfoil at various angles of attack [51]</td>
<td>58</td>
</tr>
<tr>
<td>6.6</td>
<td>Grid properties for NLR-7301 airfoil</td>
<td>60</td>
</tr>
<tr>
<td>6.7</td>
<td>Computed locations of transition onset on main element of NLR-7301 airfoil over a range of Mach numbers with ( Re = 2.2 \times 10^6 ), ( \text{Tu}_\infty = 0.30% ), and an angle of attack of 0.85°</td>
<td>62</td>
</tr>
</tbody>
</table>
# List of Figures

2.1 Scaled vorticity Reynolds number profile in Blasius boundary layer flow [21] ........... 14

3.1 Blocking strategy for a NACA0012 airfoil ......................................................... 24

5.1 Flat plate grid and geometry ................................................................. 38
5.2 NACA0012 grid and geometry ................................................................. 38
5.3 Convergence history for Case 1A1, with baseline solution algorithm and parameters . . 40
5.4 Convergence histories for Test Case 1A1, with varying $R_{\text{max,dt}}$ values and a $\Delta t_{\text{max,dt}}$ value of 10 ................................................................. 41
5.5 Convergence histories for Test Case 1A1, with varying $\Delta t_{\text{max,dt}}$ values and an $R_{\text{max,dt}}$ value of 0.01 ................................................................. 42
5.6 Comparison of convergence histories for Test Case 1A1 with time step scaling given by equation (5.2), and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 0.01, respectively ................. 43
5.7 Convergence histories for Test Case 2A1 with variable limiting methods and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, respectively ................................................................. 45
5.8 Convergence histories for Test Case 2C1 with variable clipping and adaptive reference time step methods and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, respectively ................................................................. 45
5.9 Convergence histories for Test Case 1A2 residual reduction study and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 0.01, respectively ................................................................. 46
5.10 Convergence histories for Test Case 2A2 residual reduction study and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, respectively ................................................................. 47
5.11 Convergence histories for Test Case 2C2 with negative Spalart-Allmaras model modification, $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, variable clipping, an adaptive reference time step, and an $R_d$ value of $1 \times 10^{-5}$ ................................................................. 49
5.12 Convergence histories for Test Case 2A1 with and without transition prediction, $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, respectively, variable clipping, an adaptive reference time step, and an $R_d$ value of $1 \times 10^{-5}$ ................................................................. 50

6.1 Skin-friction profiles for zero-pressure-gradient flat-plate, Medida and Baeder correlation . 52
6.2 Skin-friction profiles for zero-pressure-gradient flat-plate, Langtry correlation ............ 53
6.3 Grid and geometry for NLF0416 airfoil [50] .................................................... 54
6.4 Grid and geometry for S809 airfoil [51] ......................................................... 55
6.5 Skin-friction profiles for NLF0416 airfoil at positive angles of attack ..................... 56
6.6 Skin-friction profiles for NLF0416 airfoil at negative angles of attack ................. 57
6.7 Calculated skin-friction profiles for S809 airfoil at various angles of attack ........... 58
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.8</td>
<td>Grid and geometry for NLR-7301 airfoil [1]</td>
<td>59</td>
</tr>
<tr>
<td>6.9</td>
<td>Calculated skin-friction profiles for the main element NLR-7301 airfoil over a range of Mach numbers</td>
<td>61</td>
</tr>
<tr>
<td>6.10</td>
<td>Calculated skin-friction profiles for the trailing-edge flap of the NLR-7301 airfoil over a range of Mach numbers</td>
<td>61</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Motivation

Many conventional computational fluid dynamics (CFD) algorithms that are based on the Reynolds-averaged Navier-Stokes (RANS) equations assume that the flow is fully turbulent, i.e. that laminar-turbulent transition occurs at the leading edge of an airfoil. As a result, the important effects of laminar-turbulent transition are not included in the majority of CFD simulations. Although a significant amount of progress has been made in the development of reliable turbulence models, transition modelling does not offer the same wide spectrum of CFD-compatible models. Accurately determining the location of laminar-turbulent transition presents several difficulties. The transition process involves a wide range of length scales and is very sensitive to physical flow features such as pressure gradients and the free-stream turbulence level. In addition, transition can occur through different mechanisms depending on the application being investigated, such as natural transition, bypass transition, and separation-induced transition. The origin of turbulence and the accompanying transition from laminar to turbulent flow strongly affects flow development. It is of fundamental importance in various aerodynamic flows such as flow past wings and airframes, and has a major impact on flow characteristics including the distributions of wall shear stress and surface heat transfer, flow separation, skin friction, and viscous drag, all of which influence aerodynamic performance. Many computational studies have determined that turbulence models which assume fully turbulent flow lead to an over-prediction of drag [30]. Consequently, analysis of aerodynamic bodies using CFD packages that do not include transition prediction techniques may lead to inaccurate results. Additionally, the assumption of fully turbulent flow limits the development of natural-laminar-flow designs. The ability to accurately predict laminar-turbulent transition is particularly important for lower Reynolds number flows associated with smaller aircraft, such as unmanned aerial vehicles (UAVs). Natural-laminar-flow wings and airframes produce an increased laminar extent of the boundary layer, which results in a decrease in drag due to a smaller skin friction coefficient. In the future, the design of laminar aerodynamic bodies, which requires the ability to efficiently predict laminar-turbulent transition, will play a key role in a reduction in fuel consumption due to this decrease in drag.
1.2 Boundary-Layer Transition

There are several mechanisms for boundary-layer transition. These include natural transition, bypass transition, separation-induced transition, and transition due to crossflow instabilities or roughness. The method of transition depends greatly on the application being studied, such as flow past wings, airframes, rotor blades, and wind turbine blades, and the characteristics of the free-stream flow, such as Reynolds number, angle of attack, pressure gradient, and turbulence intensity. The location of transition onset can be identified as the location where the skin friction coefficient first begins to increase [29]. This section will discuss each method and its impact on flow dynamics.

1.2.1 Natural Transition

Natural transition is the primary method for laminar-turbulent transition in aerodynamic flows. It is characterized by the exponential growth of two-dimensional Tollmien-Schlichting (TS) waves, leading to nonlinear disturbances and eventually fully turbulent flow.

In the first region of transition, the laminar boundary layer is destabilized by the presence of viscous instability waves, TS waves [46]. When the free-stream turbulence level is low, i.e. below 1% [28], a laminar boundary layer becomes linearly unstable beyond a critical Reynolds number at which TS waves start to grow [21]. This instability is due to viscosity effects which destabilize the TS waves, causing them to slowly grow throughout the transition process, leading to a breakdown of the laminar boundary layer. This process might not be complete until a streamwise distance as large as 20 times farther downstream from the leading edge than the initial starting position of linear instability [62]. Inviscid mechanisms produce three-dimensional disturbances once the waves have become nonlinear in the form of turbulent spots. These spots grow in the surrounding laminar flow until a fully turbulent boundary layer is produced.

The natural transition process can be accelerated in the case of swept wings, where a significant velocity component in the sweep direction is present. This causes inflectional instability in the boundary layer, which can result in transition occurring further upstream [54]. This is known as crossflow instability. There are two forms of crossflow instability, stationary and traveling [9]. Stationary instabilities are produced by steady surface roughness, while unsteady sources, such as free-stream turbulence growth are responsible for traveling crossflow instabilities. In order to accurately simulate three-dimensional swept wings all three disturbances, TS waves, stationary, and travelling crossflow instabilities, must be modelled. The interactions of these three phenomena directly influence the location of transition.

1.2.2 Bypass Transition

As opposed to natural transition, bypass transition occurs at high free-stream turbulence levels, i.e. greater than 1%. It is usually associated with turbomachinery applications where upstream blade rows generate large levels of turbulence. Therefore, a free-stream turbulence intensity of 1% is often taken as the boundary between natural transition and bypass transition. For bypass transition, transition occurs through the diffusion of free-stream turbulence into the boundary layer. This results in the first stage, and up to second and third stages, of natural transition being bypassed. Currently, researchers have been unable to detect TS waves when the free-stream turbulence level is greater than 1% [28]. Nonlinear instabilities form directly in the laminar boundary layer, leading to the formation of turbulent spots. Alternatively, surface roughness can create disturbances at the wall, which can lead to bypass transition.
In the case of bypass transition, the flow before the transition location is fully laminar, as opposed to natural transition where flow disturbances, such as TS waves, are evident before this point. As with natural transition, the development of these turbulent spots eventually leads to the formation of fully turbulent flow.

1.2.3 Separation-Induced Transition

Another important method for transition is separation-induced transition. This occurs when a laminar boundary layer separates in the presence of a strong adverse pressure gradient. Transition may occur in the separated shear layer, developing a laminar-separation/turbulent-reattachment bubble [28]. This transition process can include many of the steps involved in natural transition. The length of this bubble is dependent on the Reynolds number, angle of attack, surface roughness, and turbulence intensity. Separation bubbles are usually classified as being either long or short. Short separation bubbles affect the local pressure distribution and can be useful for tripping the boundary layer to a fully turbulent profile. Turbulent boundary-layer profiles allow for larger adverse pressure gradients before the aerodynamic body encounters stalling, as turbulent boundary layers are able to sustain an adverse pressure gradient and prevent flow separation better than an equivalent laminar boundary layer. Long separation bubbles influence the overall pressure distribution and can generate significant pressure losses. Depending on flow conditions, long separation bubbles can develop from short bubbles and vice versa, this process is known as bursting [29]. The sudden change from a short to long separation bubble can result in a drastic loss of lift and cause the airfoil to stall if the separated flow does not reattach.

1.2.4 Relaminarisation

Turbulent boundary-layer flow can undergo relaminarisation, or reverse transition, in the presence of a favourable pressure gradient, such as if the flow is strongly accelerated. This often occurs on the leading edge of the suction side of a turbine or near the trailing edge of the pressure side of an airfoil. Downstream of this relaminarisation it is possible for the flow to transition back to turbulent flow.

1.3 Transition Prediction and Modelling Techniques

There are several difficulties which need to be overcome in the development a CFD-compatible transition model. The majority of transition modelling techniques apply non-local operations, such as the integration of the boundary layer, which limit their implementation in modern CFD codes. In their work, Pasquale and Rona [37] specify eight requirements for a fully CFD-compatible transition model:

1. Provide the calibrated prediction of transition onset and length
2. Include different transition mechanisms
3. Be formulated locally
4. Avoid multiple solutions
5. Do not affect underlying turbulence model in fully turbulent regimes
6. Similar convergence as the underlying turbulence model
7. Independent of coordinate system

8. Applicable to three-dimensional boundary layers

The following sections will review common transition modelling techniques based on these criteria.

1.3.1 Direct Numerical Simulations and Large-Eddy Simulations

As the name implies, Direct Numerical Simulations (DNS) solve the time-dependent Navier-Stokes equations directly. Since there is no Reynolds averaging, there is no requirement for turbulence closure by a turbulence model. Therefore DNS simulations are able to produce all of the mechanisms of boundary-layer transition, such as laminar flow breakdown, the development of turbulent spots, and the transition to fully turbulent flow [63]. However, a very fine computational grid is required in order to capture the small scales of turbulence. Specifically, a three-dimensional DNS requires a number of mesh points, \( N^3 \), satisfying \( N^3 \geq Re^{9/4} \), where \( Re \) is the turbulent Reynolds number [6]. Furthermore, the number of time steps required grows as a power law of the Reynolds number. Based on the number of mesh points and the number of time steps, the number of operations required to complete a simulation can be estimated as \( Re^3 \) [6]. This currently leads to impractical computational costs, at all but very low Reynolds numbers. Consequently, DNS simulations are limited to research applications, such as the studying of transition mechanisms, rather than engineering applications, which require the solution of large and complex geometries.

Large-Eddy Simulations (LES) were developed in order to reduce the computational cost associated with DNS simulations. Similar to DNS, LES computations solve the Navier-Stokes equations directly. However, while the large scale turbulent eddies are fully resolved, the small scale eddies are modelled using an eddy-viscosity approach [49]. Although more computationally efficient, the location of transition has been shown to be dependent on the Smagorinsky constant specified to calibrate the subgrid eddy viscosity [12]. Germano implemented a method to determine this constant locally [12]; however work performed by Michelassi et al. [32] demonstrated noticeable differences in the quantitative comparison of DNS and LES simulations for transitional flows. As with DNS simulations, LES computations require substantial computational costs making them more suitable for research applications.

1.3.2 \( \epsilon^N \) Method

The \( \epsilon^N \) method for predicting transition is the lowest closure level where the instability of the flow is simulated [21]. It is a semi-empirical approach, with the \( N \)-factor depending on free-stream flow conditions, and is based on linear stability theory. Linear stability theory involves the study of the behaviour of small disturbances in the flow, and determining whether they will grow or not. Transition occurs when the disturbance amplitude grows, indicating unstable flow. The \( \epsilon^N \) transition model is able to accurately simulate natural transition due to TS waves. However, because it is based on linear theory, the model is unable to predict transition due to nonlinear interactions, such as high free-stream turbulence intensity and surface roughness. The model is therefore unable to predict bypass or separation-induced transition [21]. In addition, although \( \epsilon^N \) methods are effective at predicting the onset of transition, the transitional regime itself is not modelled. Moreover, \( \epsilon^N \) methods are very non-local, requiring a large infrastructure to apply the code. However, recent work by Krimmelbein et al. [16, 17] has demonstrated that transition models based on \( \epsilon^N \) methods can be parallelized and extended to three-dimensional flows.
There are two options for completing a simulation using $e^N$ and other nonlocal methods. A simulation can be conducted with a RANS CFD solver coupled with a boundary-layer code in order to obtain the laminar velocity and temperature profiles along the body of interest, or the boundary-layer properties can be obtained from the Navier-Stokes solution directly. A boundary-layer code is often used to increase the accuracy in determining the boundary-layer edge, which is required to satisfy the requirements of the stability analysis [54]. Boundary-layer solvers are able to accurately calculate the boundary-layer thickness on coarse grids, reducing the computational cost of simulations; however, the integration of a boundary-layer solver introduces several disadvantages. These include complications when attempting to couple the two solvers, redundancy in solving both the RANS and boundary-layer equations, and the restriction to modelling fully-attached to mildly separated flows [39]. Recent work conducted by Rashad and Zingg [39] has demonstrated the benefit of defining the boundary-layer edge based on the Navier-Stokes solution, rather than using a boundary-layer solver. Using a structured mesh, they were able to accurately compute the laminar boundary-layer properties using several boundary-layer edge-finding methods, avoiding the complications of implementing a boundary-layer solver. Furthermore, Krumbein et al. [18, 19, 20] have demonstrated the effectiveness of using hybrid grids with an unstructured grid in the free stream and a structured grid in the boundary layer, which facilitates the integration of the boundary-layer properties while simplifying the grid generation process.

A stability analysis method is applied to determine the local growth rates of the unstable waves for each boundary-layer profile. These local growth rates are integrated along the streamlines of the flow in order to determine the $N$ factor, which is used to calculate the disturbance amplitude ratio, $e^N$. The location of transition onset is defined when this value exceeds the limiting $N$ factor. The stability analysis conducted in this process can be provided by solving the local stability equations, or the Parabolized Stability Equations (PSE) [21]. The PSE equations are a set of nonlinear stability equations which are able to simulate the development of unstable waves up to the weakly nonlinear regime [44]. Therefore, $e^N$ methods using the PSE equations provide the capability to predict transition due to nonlinear effects, such as bypass transition. However, this approach is still not able to predict separation-induced transition, which requires the modelling of stronger nonlinear interactions [44].

The calculation of stability analysis equations can be avoided altogether using the simplified $e^N$ envelope method, developed by Drela [11], and used by Rashad and Zingg [39] and Mayda [27]. This method makes direct use of the boundary-layer properties to approximate the envelope of the spatial amplification of the disturbances, i.e. the $N$ factors. The boundary-layer properties are related to the $N$-factor using a set of correlations developed based on linear stability results of the Falkner-Skan family of velocity profiles, and although the envelope method does not track individual frequencies, it has been shown to be more efficient than solving stability equations directly [27]. Reference [39] provides further information on the simplified $e^N$ method.

Transition models based on the $e^N$ method and linear stability analysis are effective due to their ability to capture the physics of transition onset, but their non-local formulation and complexity make them difficult to incorporate in general-purpose CFD packages.

### 1.3.3 Low-Reynolds-Number Turbulence Models

Low-Reynolds-number (Low-Re) transition models are eddy-viscosity models, such as the SA and $k - \omega$ turbulence models, which are adapted to compute low Reynolds number flow. The work performed by Wilcox [61] explains the ability of the $k - \omega$ model to describe transition due to the properties of the
production and destruction terms. Low-Re transition models have historically been the most compatible with modern CFD methods, as they require little alteration to the existing framework; however, low-Re transition models have been shown to suffer from a close interaction between viscous sublayer damping and transition prediction [42, 43]. This results in difficulties when trying to calibrate the model to accurately simulate either phenomenon. Eddy-viscosity turbulence models are originally calibrated for viscous sublayer damping, and re-calibration to increase transition prediction will affect the performance of this damping. Therefore, although low-Re models are compatible with modern CFD framework their poor accuracy prevents them from being used as a reasonable transition prediction method.

1.3.4 Empirical Correlations

Empirical correlations are often viewed as the engineering alternative to other transition models, as they are relatively easy to calibrate and are sufficiently accurate to capture the major transition mechanisms. Furthermore, depending on the application, correlations can be developed to model the different transition mechanisms, which include natural transition, bypass transition, or transition due to crossflow instabilities and roughness. The empirical correlations developed by Abu-Ghannam and Shaw [2], Mayle [28], and Suzen et al. [57] are the most widely adopted. These correlations relate the non-local transition momentum thickness Reynolds number to local quantities such as the turbulence intensity and pressure gradient parameter. The transition momentum thickness Reynolds number is calculated using the empirical correlations, and compared to the actual momentum Reynolds number determined from a laminar computation of the boundary layer. This determines the transition onset location; however, it does not provide any information about the transitional regime. Overall, empirical correlations are a promising method for predicting transition due to their ease of calibration and accuracy; however their non-local operations limit their implementation in general-purpose CFD codes.

Langtry and Menter [21] developed an approach for a correlation-based transition model, using the empirical correlations developed by Abu-Ghannam and Shaw [2], which does not require non-local information of the flow. The transition model formulated by Langtry is composed of two transport equations. The first is an equation for the intermittency, $\gamma$, which is used to trigger the turbulence transition process by controlling the level of turbulence in the boundary layer. The second is an equation for the transition momentum thickness Reynolds number, $Re_{th}$, which in the free stream is equal to the value produced by the empirical correlations, $Re_{\theta t}$, and is diffused into the boundary layer using a standard diffusion term. The purpose of this equation is to transform a non-local empirical correlation into a local value, which can then be compared with the local vorticity Reynolds number, $Re_{\nu}$. The use of the vorticity Reynolds number eliminates the need to calculate the momentum thickness Reynolds number, $Re_{\theta}$, and enables the flow to be calculated using the local values of density, viscosity, wall distance, and vorticity. At each location in the flow where the vorticity Reynolds number exceeds the transition onset Reynolds number, a source term in the intermittency equation is activated and turbulence is produced.

In Langtry’s method the transition model equations are coupled with the $k - \omega$ SST turbulence model [21]. However, the model constants in the Spalart-Allmaras (SA) turbulence model [52] are tuned for external aerodynamic flows, and the model is shown to give good performance in boundary layers with adverse pressure gradients, which is important for predicting flow separation and laminar-turbulent transition [59]. In addition, the SA turbulence model consists of one transport equation for an eddy-viscosity-like variable, which reduces computational cost relative to the two-equation SST turbulence
model. Therefore, in order to increase the efficiency and accuracy of the flow solver, Medida and Baeder [30] modified the empirical correlation-based transition model to be compatible with the SA turbulence model. Further work by Schucker [47], and Suluksna et al. [56] in coupling the transition model with the SA turbulence model was conducted, including additional calibration of the empirical correlations. Validation studies were conducted for the transition model, coupled with each turbulence model, and compared with experimental results to ensure accuracy of the calibrations [25, 30]. However, an investigation by Aranake et al. [4], evaluating the performance of the three-equation $\gamma - Re_{\theta t}$ SA transition model, discovered that the transition model requires additional calibration for flows involving transition due to laminar flow separation. In addition, the model has not been extended to include cross-flow instabilities and high-speed flow correlations. However, empirical correlations for such flows have been developed in more recent work by Medida and Baeder [31], and can be implemented into the current transition model. The work of Coder and Maughner [8] builds off the work of Langtry and Menter [21]; however, the $\tilde{Re}_{\theta t}$ equation is replaced with an $\epsilon^N$ method for transition prediction, which is based on linear stability theory.

1.4 Flow Solver

The flow solver used in the current work is a three-dimensional multi-block structured finite-difference solver developed by Osusky and Zingg [36]. The governing equations are spatially discretized using summation-by-parts operators, with simultaneous approximation terms applied to enforce boundary conditions and inter-block coupling. To decrease the computational time required to complete a flow solution, the computational domain is decomposed into multiple blocks, resulting in multi-block structured grids, which allow for parallel computation of the domain. An implicit Newton-Krylov solution algorithm making use of a pseudo-transient continuation (PTC) strategy is applied to the set of nonlinear algebraic equations produced by the spatial discretization of the governing equations to drive the solution from an initial guess to a converged steady-state solution. The large system of linear equations generated at each iteration of the PTC strategy is solved using the GMRES linear solver [40], preconditioned using an approximate-Schur parallel preconditioner [41].

1.5 Thesis Objectives and Outline

The purpose of this work is to investigate the performance of a two-equation $\gamma - \tilde{Re}_{\theta t}$ transition model, based on differential transport equations formulated with empirical correlations coupled to the SA turbulence model in a three-dimensional multi-block structured finite-difference solver [36]. It is inspired by the work of Langtry and Menter [21, 23, 25, 22, 24] in developing a $\gamma - \tilde{Re}_{\theta t}$ transition model coupled to the SST $k - \omega$ turbulence model, and the efforts of Schucker [47], and Medida and Baeder [30] in adapting this model to be coupled with the one equation SA turbulence model [52]. The goal is to determine whether the algorithm can accurately and efficiently predict transition in two-dimensional flows due to natural transition and separation-induced transition when implemented in this multi-block structured finite-difference solver.

The remainder of this work is organized as follows. Chapter 2 provides a summary of the governing equations, including the transition model equations, which are spatially discretized based on the methods presented in Chapter 3. Chapter 4 describes the solution strategy utilized to iterate the spatially
discretized equations to steady state. Chapter 5 investigates the accuracy and efficiency of the algorithm, and presents methods for optimizing the algorithm and their impact on accuracy and efficiency. Results for two and three-dimensional test cases are examined in Chapter 6, while Chapter 7 provides conclusions, contributions, and recommendations.
Chapter 2

Governing Equations

This chapter presents the governing equations for the simulations in this thesis. These include the three-dimensional compressible Navier-Stokes equations, the one-equation Spalart Allmaras turbulence model, and the two-equation $\gamma - Re_{\theta t}$ transition model. The formulation of the transition model is described here, as well as the curvilinear coordinate transformation of these equations from Cartesian space into computational space. The boundary and initial conditions for the transition model are introduced as well. The curvilinear coordinate transformation, boundary conditions, and initial conditions for the Navier-Stokes equations and the Spalart-Allmaras turbulence model can be found in the work produced by Osusky and Zingg [35], which provides the framework for the current work.

2.1 Navier-Stokes Equations

The compressible Navier-Stokes equations consist of a set of five partial differential equations that represent conservation of mass, momentum, and energy. In Cartesian coordinates and conservative form the three-dimensional Navier-Stokes equations, and the associated non-dimensional quantities, are specified as

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = \frac{1}{Re} \left( \frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y} + \frac{\partial G_v}{\partial z} \right),$$

and

$$t = \frac{\tilde{a}_\infty}{c}, \quad x = \frac{x}{c}, \quad y = \frac{y}{c}, \quad z = \frac{z}{c}, \quad \rho = \frac{\rho}{\rho_\infty}, \quad u = \frac{u}{a_\infty}, \quad v = \frac{v}{a_\infty}, \quad w = \frac{w}{a_\infty}, \quad \mu = \frac{\mu}{\mu_\infty}, \quad e = \frac{e}{\rho_\infty a_\infty^2}.$$  

The pressure and molecular viscosity are determined using the non-dimensional form of the equation of state for an ideal gas and Sutherland’s law, respectively, which are given by

$$p = (\gamma - 1) \left( e - \frac{1}{2} \rho (u^2 + v^2 + w^2) \right),$$

and
\[ \mu = \frac{a^3(1 + \frac{S^*}{T^\infty})}{a^2 + \frac{S^*}{T^\infty}}. \] (2.3)

The Cartesian coordinates are represented as \((x, y, z)\), while the Cartesian velocity components are given by \((u, v, w)\). The remainder of the variables are defined as follows: \(\rho\) is the density, \(a\) the speed of sound for an ideal gas calculated using \(\sqrt{\frac{\gamma p}{\rho}}\), \(e\) the total energy per unit volume, \(c\) the characteristic length, \(\gamma\) the specific heat ratio, \(\mu\) the molecular viscosity, \(p\) the pressure, \(S^*\) Sutherland’s constant \((196^\circ R)\), and \(T^\infty\) the free-stream temperature, taken as \((460^\circ R)\) for air. Free-stream values are denoted by ‘\(\infty\)’, while ‘\(\tilde{\cdots}\)’ represents a dimensional quantity. The Reynolds number is defined as

\[ \text{Re} = \frac{\rho \infty a \infty c \infty}{\mu \infty}. \] (2.4)

The non-dimensional conservative variables and inviscid flux vectors are given by

\[
Q = \begin{bmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
e
\end{bmatrix}, \quad E = \begin{bmatrix}
\rho u \\
\rho u^2 + p \\
\rho uv \\
\rho uw \\
ue + p
\end{bmatrix}, \quad F = \begin{bmatrix}
\rho v \\
\rho v^2 + p \\
\rho vw \\
\rho wz \\
v(e + p)
\end{bmatrix}, \quad G = \begin{bmatrix}
\rho w \\
\rho uw \\
\rho vw \\
\rho wz \\
w(e + p)
\end{bmatrix}.
\] (2.5)

The viscous flux vectors are defined as

\[
E_v = \begin{bmatrix}
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
E_{v,5}
\end{bmatrix}, \quad F_v = \begin{bmatrix}
0 \\
\tau_{yx} \\
\tau_{yy} \\
\tau_{yz} \\
F_{v,5}
\end{bmatrix}, \quad G_v = \begin{bmatrix}
0 \\
\tau_{zx} \\
\tau_{zy} \\
\tau_{zz} \\
G_{v,5}
\end{bmatrix},
\] (2.6)

where the viscous stress terms are denoted as

\[
\begin{align*}
\tau_{xx} &= \frac{4}{3}(\mu + \mu_t)(u_x) - \frac{2}{3}(\mu + \mu_t)(v_y + w_z), \\
\tau_{xy} &= (\mu + \mu_t)(u_y + v_x), \\
\tau_{xz} &= (\mu + \mu_t)(u_z + w_x), \\
\tau_{yy} &= \frac{4}{3}(\mu + \mu_t)(v_y) - \frac{2}{3}(\mu + \mu_t)(u_x + w_z), \\
\tau_{yz} &= (\mu + \mu_t)(v_z + w_y), \\
\tau_{zz} &= \frac{4}{3}(\mu + \mu_t)(w_z) - \frac{2}{3}(\mu + \mu_t)(u_x + v_y),
\end{align*}
\] (2.7)

and the heat conduction terms by

\[
\begin{align*}
E_{v,5} &= u\tau_{xx} + v\tau_{xy} + w\tau_{xz} + (\mu Pr^{-1} + \mu_t Pr_t^{-1})(\gamma - 1)^{-1}\partial_x(a^2), \\
F_{v,5} &= u\tau_{yx} + v\tau_{yy} + w\tau_{yz} + (\mu Pr^{-1} + \mu_t Pr_t^{-1})(\gamma - 1)^{-1}\partial_y(a^2), \\
G_{v,5} &= u\tau_{zx} + v\tau_{zy} + w\tau_{zz} + (\mu Pr^{-1} + \mu_t Pr_t^{-1})(\gamma - 1)^{-1}\partial_z(a^2).
\end{align*}
\] (2.8)
The turbulent eddy viscosity is represented as $\mu_t$, and $Pr_t$ and $Pr$ are the laminar and turbulent Prandtl numbers, defined as 0.72 and 0.90, respectively.

## 2.2 Spalart-Allmaras Turbulence Model

Turbulent flow can be simulated by adding a turbulent or eddy viscosity term, $\mu_t$, which is solved for using a turbulence model, to the molecular viscosity, $\mu$. In the original $\gamma - Re_{\theta t}$ transition model, developed by Langtry [21], the transition model equations are coupled with the $k - \omega$ SST turbulence model. However, the model constants in the Spalart-Allmaras turbulence model are tuned for external aerodynamic flows [58, 48], and the model has been shown to give good performance in boundary layers with adverse pressure gradients, which is important for predicting flow separation and laminar-turbulent transition [59].

The Spalart-Allmaras turbulence model solves a partial differential equation for an eddy-viscosity-like variable $\tilde{\nu}$. In non-dimensional form, the model equation is defined as

$$
\frac{\partial \tilde{\nu}}{\partial t} + u \frac{\partial \tilde{\nu}}{\partial x} + v \frac{\partial \tilde{\nu}}{\partial y} + w \frac{\partial \tilde{\nu}}{\partial z} = P_\nu - D_\nu + \frac{1 + c_{k2}}{\sigma Re} \nabla \cdot [(\nu + \tilde{\nu}) \nabla \tilde{\nu}] - \frac{c_{k2}}{\sigma Re} (\nu + \tilde{\nu}) \nabla^2 \tilde{\nu} \tag{2.9}
$$

The terms on the left-hand side represent advection, the second and third terms on the right side simulate diffusion, while $P_\nu$ and $D_\nu$ represent the production and destruction terms given by

$$
P_\nu = \frac{c_{k1}}{Re} [1 - f_{t2}] \tilde{\nu} \tilde{\nu},
$$

and

$$
D_\nu = \frac{1}{Re} \left[ c_{w1} f_w - \frac{c_{k1}}{k^2} f_{t2} \right] \left( \frac{\tilde{\nu}}{d} \right)^2,
$$

where $d$ is the distance to the closest solid surface. The viscosity like variable $\tilde{\nu}$, normalized with $\tilde{\nu}_\infty$, is related to the kinematic eddy viscosity $\nu_t$ as

$$
\nu_t = \tilde{\nu} f_{v1}, \tag{2.10}
$$

where

$$
f_{v1} = \frac{\chi^3}{\chi^3 + \chi^4_{v1}}, \tag{2.11}
$$

and

$$
\chi \equiv \frac{\tilde{\nu}}{\nu}, \tag{2.12}
$$

and the molecular and turbulent kinematic viscosities are

$$
\nu = \frac{\mu}{\rho}, \tag{2.13}
$$

with
\[ \nu_t = \frac{\mu_t}{\rho}. \quad (2.14) \]

The \( f_w \) term is given by
\[ f_w = g \left[ 1 + c_{w2}^6 \right]^{\frac{\hat{\nu}}{\hat{\sigma}}}, \quad (2.15) \]
where
\[ g = r + c_{w2}(r^6 - r), \quad (2.16) \]
and
\[ r = \min \left( \frac{\sigma}{S k^2 d^2}, 10 \right). \quad (2.17) \]

The modified vorticity value, \( \hat{S} \), normalized using \( \hat{\rho}_\infty \hat{c}_1 \), and the vorticity magnitude, \( S \), normalized using \( \frac{S_{\infty}}{c} \), are defined as
\[ \hat{S} = ReS + \frac{\hat{\nu}}{k^2 d^2} f_{v2}, \quad (2.18) \]
and
\[ S = \left[ \left( \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right)^2 + \left( \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)^2 \right]^{-\frac{1}{2}}, \quad (2.19) \]
where \( f_{v2} \) is given by,
\[ f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}. \quad (2.20) \]

The constants are defined as
\[
\begin{align*}
  c_{b1} &= 0.1355, & c_{b2} &= 0.622, \\
  \sigma &= 2/3, & c_{v1} &= 7.1, \\
  k &= 0.41, & c_{t1} &= 5.0, \\
  c_{t2} &= 2.0, & c_{t3} &= 1.2, \\
  c_{t4} &= 0.5, & c_{w1} &= \frac{c_{b1}}{k^2} + \frac{1}{\sigma}(1 + c_{b2}), \\
  c_{w2} &= 0.3, & c_{w3} &= 2.0.
\end{align*}
\]

The Spalart-Allmaras turbulence model has the capability to be expanded to include the specification of an explicit laminar-to-turbulent transition location in the form of two trip terms; however, this has been omitted for the current work, where the goal is to study the performance of the transition model.
2.3 \( \gamma-Re_{\theta t} \)-Spalart-Allmaras Model

The transition model employed in this thesis is a local empirical correlation-based transition model, originally developed by Langtry [21] for use with the two-equation \( k - \omega \) SST turbulence model. As mentioned in Section 1.3.4, the model consists of two transport equations. The first is for the intermittency, \( \gamma \), which is used to trigger the transition process, and the second is for the transition momentum thickness Reynolds number, \( Re_{\theta t} \), which describes the physics of the transition process. The following sections provide a background on the formulation of the transition model developed by Langtry, its adaptation to be compatible with the SA turbulence model by Medida and Baeder [30] and Schucker [47], along with the curvilinear coordinate transformation to computational space, boundary conditions, and initial conditions.

2.3.1 Model Formulation

Vorticity Reynolds Number

Traditionally, empirical correlation-based transition models predict transition onset using the momentum thickness Reynolds number, \( Re_{\theta} \). However, instead of using the momentum thickness Reynolds number, which is a non-local quantity, the transition model developed by Langtry [21] determines the onset of transition using the vorticity Reynolds number, defined as

\[
Re_{\nu} = \frac{\rho d^2 S}{\mu}.
\]

(2.21)

The vorticity Reynolds number is calculated using only local quantities and therefore can be computed at every point in the flow. The vorticity Reynolds number is scaled in order to have a maximum value of unity in Blasius boundary layer flow, as illustrated in Figure 2.1. The scaling is achieved by dividing the vorticity Reynolds number profile in a Blasius boundary layer by the momentum thickness Reynolds number, and a calibration constant of 2.193. Specifically, the maximum value of the vorticity Reynolds number is related to the momentum thickness Reynolds number by

\[
Re_{\theta} = \frac{\max(Re_{\nu})}{2.193}.
\]

(2.22)

The transition criterion for the current work is based on this relationship, which provides the framework to compute the transition location locally. This relationship is described in further detail in the work by Langtry [21].

Empirical Correlations

The transport equations for the transition model do not attempt to model the transition process. Rather, the empirical correlations supply the physics for the transition process, while the transport equations provide the framework for their implementation. Therefore, the correlations being used determine the accuracy of the simulations. Furthermore, depending on the application being studied, correlations can be developed to model the different methods for laminar turbulent transition, such as natural, bypass, or separation-induced transition. The empirical correlations used in this model are based on the correlations developed by Abu-Ghannam and Shaw [2], which were subsequently adapted by Langtry and Menter [23, 25, 22, 24], and further refined by Langtry [21].
The empirical correlations are calculated using the free-stream turbulence intensity, $T_{u\infty}$, and the pressure gradient parameter, $\lambda$. The original transition model developed by Langtry was coupled with the two-equation $k-\omega$ SST turbulence model. This allowed for the turbulence intensity to be calculated locally using the turbulent kinetic energy $k$. However, it was shown in the work performed by Suluksna et al. [56] that the local variation in turbulence intensity is a direct consequence of a non-zero pressure gradient acting on the boundary layer. Therefore, it is redundant to use the local turbulence intensity value in addition to the pressure gradient parameter. Since the focus of this work is to couple the $\gamma$ - $Re_{\theta t}$ transition model with the SA turbulence model, where a local value of $T_u$ cannot be obtained, the free-stream $T_u$ value will be used along with the locally determined pressure gradient parameter. This method was first introduced in the work performed by Medida and Baeder [30]. The local value of the pressure gradient parameter can be calculated using

$$\lambda = \frac{\rho \theta U^2}{\mu} \frac{dU}{ds},$$  \hspace{1cm} (2.23)$$

where $dU/ds$ is the acceleration along the streamwise direction. It is defined as

$$\frac{dU}{ds} = \left[ \left( \frac{u}{U} \right) \frac{dU}{dx} + \left( \frac{v}{U} \right) \frac{dU}{dy} + \left( \frac{w}{U} \right) \frac{dU}{dz} \right].$$  \hspace{1cm} (2.24)$$

The velocity magnitude is calculated using

$$U = (u^2 + v^2 + w^2)^{\frac{1}{2}},$$  \hspace{1cm} (2.25)$$
while the derivatives of the velocity magnitude in the $x$, $y$, and $z$ directions are given by

\[
\frac{dU}{dx} = (u^2 + v^2 + w^2)^{-\frac{1}{2}} \cdot \left[ \frac{du}{dx} + v \frac{dv}{dx} + w \frac{dw}{dx} \right],
\]
\[
\frac{dU}{dy} = (u^2 + v^2 + w^2)^{-\frac{1}{2}} \cdot \left[ \frac{du}{dy} + v \frac{dv}{dy} + w \frac{dw}{dy} \right],
\]
\[
\frac{dU}{dz} = (u^2 + v^2 + w^2)^{-\frac{1}{2}} \cdot \left[ \frac{du}{dz} + v \frac{dv}{dz} + w \frac{dw}{dz} \right].
\]

The empirical correlation is defined as

\[
Tu \leq 1.3, \quad Re_{\theta_1} = \left[ 1173.51 - 589.428Tu + \frac{0.2196}{Tu^2} \right] F(\lambda_\theta),
\]
\[
Tu > 1.3, \quad Re_{\theta_1} = 331.50[Tu - 0.5658]^{-0.671} F(\lambda_\theta),
\]
\[
\lambda_\theta \leq 0, \quad F(\lambda_\theta) = 1 - [-12.986\lambda_\theta - 123.66\lambda_\theta^2 - 405.68\lambda_\theta^3]e^{-\left(\frac{Tu}{1.5}\right)^{1.5}},
\]
\[
\lambda_\theta > 0, \quad F(\lambda_\theta) = 1 + 0.275[1 - e^{-35\lambda_\theta}]e^{-\left(\frac{Tu}{1.5}\right)^{1.5}},
\]

where the transition momentum Reynolds number is denoted as

\[
Re_{\theta_1} = \frac{\rho U \theta}{\mu}.
\]

The empirical correlation is an implicit equation for the boundary-layer momentum thickness, $\theta$, as it appears on both sides of the empirical correlation equation. It appears in the transition momentum thickness Reynolds number $Re_{\theta_1}$, and the pressure gradient parameter $\lambda_\theta$. In the current work $\theta$ in equations 2.30-2.33 is solved for iteratively using Newton’s method.

**Transport Equation for Intermittency**

As mentioned in Section 1.3.4, the empirical correlation determines the transition onset location; however it does not provide any information about the transitional regime. This is supplied using the concept of intermittency, which can be defined as the fraction of the time the flow is turbulent relative to the total time. A value of zero indicates fully laminar flow, while a value of unity represents fully turbulent flow. Although the original model for intermittency, developed by Dhawan and Narasimha [10], consisted of an algebraic formulation for $\gamma$, Langtry states an advantage of using a transport equation consisting of diffusion, advection, and source terms is that intermittency can be solved throughout the boundary layer, and not just in the flow direction [21].

The transport equation for intermittency is given in non-dimensional form as

\[
\frac{\partial (\rho \gamma)}{\partial t} + \frac{\partial (\rho u \gamma)}{\partial x} + \frac{\partial (\rho v \gamma)}{\partial y} + \frac{\partial (\rho w \gamma)}{\partial z} = P_\gamma - D_\gamma + \frac{1}{Re} \nabla \left( \mu + \frac{\mu_t}{\sigma_f} \right) \nabla \gamma.
\]

The production term is defined as

\[
P_\gamma = F_{\text{length}} c_{a1} \rho s[\gamma F_{\text{onset}}]^{\frac{1}{2}}(1 - c_{e1} \gamma),
\]
where $s$ is the strain rate magnitude, given by

$$s = \sqrt{2s_{ij}s_{ij}},$$  
(2.36)

$$s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right).$$  
(2.37)

The production term is designed to be inactive in the laminar boundary layer and to trigger the growth of intermittency, and therefore turbulence, once transition is detected. This process is controlled by the two functions $F_{\text{onset}}$ and $F_{\text{length}}$, which determine the location of transition onset and the length of the transition region, respectively. $F_{\text{onset}}$ is designed to switch rapidly from a value of zero in the laminar boundary layer to a value of unity wherever the local vorticity Reynolds number exceeds the local transition onset criterion. The function is given by

$$F_{\text{onset}} = \max(F_{\text{onset2}} - F_{\text{onset3}}, 0),$$  
(2.38)

$$F_{\text{onset3}} = \max\left(1 - \left(\frac{R_T}{2.5}\right)^3, 0\right),$$  
(2.39)

$$F_{\text{onset2}} = \min(\max(F_{\text{onset1}}, F_{\text{onset1}}^4), 2),$$  
(2.40)

$$F_{\text{onset1}} = \frac{Re\nu}{2.193Re_{\theta c}},$$  
(2.41)

where the eddy viscosity ratio is defined as

$$R_T = \frac{\mu_t}{\mu},$$  
(2.42)

and $Re_{\theta c}$ is the critical momentum thickness Reynolds number, which represents the location where the intermittency first begins to grow in the laminar boundary layer. This value is less than the transition momentum thickness Reynolds number, because intermittency begins to grow before a change in the laminar boundary layer profile is evident [21]. Therefore, $Re_{\theta c}$ can be thought of as the point at which intermittency beings to grow, while $Re_{\theta t}$ is the location where the velocity profile begins to develop from a purely laminar to a transitional profile. The relationship between the two Reynolds number is given by

$$Re_{\theta c} = f(\tilde{Re}_{\theta t}),$$  
(2.43)

where $\tilde{Re}_{\theta t}$ is the transported value of the transition momentum thickness Reynolds number, given by equation (2.48). The value for this correlation was determined using a series of numerical experiments of a flat plate. The critical Reynolds number was varied, along with the turbulence intensity, and the transition Reynolds number was measured where the skin friction began to increase. The correlation developed by Langtry [21], and used in the work performed by Schucker [47], is defined as

$$Re_{\theta c} = (39.35 Tu_\infty^3 - 27.38 Tu_\infty^2 + 4.99 Tu_\infty + 0.543)\tilde{Re}_{\theta t}.$$  
(2.44)

However, for simulations where natural transition is the primary method of transition, and where the turbulence intensity is much less than unity, Medida and Baeder [30] found the following correlation,

$$Re_{\theta c} = (4.45 Tu_\infty^3 - 5.7 Tu_\infty^2 + 1.37 Tu_\infty + 0.585)\tilde{Re}_{\theta t},$$  
(2.45)
produced more accurate results. For the current work, where the focus of the simulations involves natural transition, the correlation developed by Medida and Baeder [30] is used. The justification for this choice is provided in Chapter 6.

The correlation for the length function, which determines the length of the transition region through controlling the magnitude of the production term, was similarly determined using numerical experiments of flat plates. However, the work performed by Schucker [47] identified that, for values of $\tilde{Re}_{\theta t}$ greater than 596, setting the $F_{\text{length}}$ function to a constant value of 0.2 resulted in transition profiles that matched the experimental values determined by Schubauer and Klebanoff [45], and Schubauer and Skramstad [46]. This method was adopted in the current work, since the values of the free-stream turbulence intensity and pressure gradient parameter generated in the simulations produce $\tilde{Re}_{\theta t}$ values larger than 596. The final term in the intermittency production term limits the maximum value of intermittency, which is controlled by the constant $c_{e1}$.

The destruction term for the intermittency transport equation is defined as

$$D_\gamma = c_{a2} \rho \bar{S}_\gamma F_{\text{turb}}(c_{e2} \gamma - 1).$$ (2.46)

This term is responsible for ensuring that $\gamma$ remains close to zero in the laminar boundary layer, and allows the flow to relaminarise when the $F_{\text{onset}}$ function, equations (2.39 – 2.42), produces values close to zero. The purpose of the function $F_{\text{turb}}$ is to determine when the flow is outside the laminar boundary layer, or in the viscous sublayer, and deactivate the destruction term. The constant $c_{a2}$ controls the magnitude of the destruction equation, while the final term limits the minimum value of intermittency, which is controlled by the constant $c_{e2}$.

The constants for the intermittency equation are defined as follows:

$$c_{e1} = 1.0, \quad c_{e2} = 50,$$
$$c_{a1} = 2.0, \quad c_{a2} = 0.06,$$
$$\sigma_f = 1.0.$$

Transport Equation for Transition Momentum Thickness Reynolds Number

The empirical correlations determine the value of the transition momentum thickness Reynolds number based on free-stream variables, specifically the turbulence intensity and pressure gradient parameter. However, the intermittency transport equation requires the local value of $\tilde{Re}_{\theta t}$ in the boundary layer, in order to calculate the transition onset $F_{\text{onset}}$ and length $F_{\text{length}}$ functions. Therefore, in the boundary layer determining the value of $\tilde{Re}_{\theta t}$ is a non-local operation. The purpose of the transition momentum thickness Reynolds number transport equation is to transform the non-local empirical correlation into a local quantity which can be calculated in the boundary layer. The transport equation is designed to calculate the value of $\tilde{Re}_{\theta t}$ in the free-stream using an empirical correlation, which is then convected and diffused into the boundary layer.

The transport equation is given by

$$\frac{\partial(\rho \tilde{Re}_{\theta t})}{\partial t} + \frac{\partial(\rho u \tilde{Re}_{\theta t})}{\partial x} + \frac{\partial(\rho v \tilde{Re}_{\theta t})}{\partial y} + \frac{\partial(\rho w \tilde{Re}_{\theta t})}{\partial z} = P_{\theta t} + \frac{1}{Re} \left[\nabla(\sigma_{\theta t}(\mu + \mu_t)\nabla \tilde{Re}_{\theta t})\right],$$ (2.47)
where the production term is defined as,

\[ P_{\theta t} = c_{\theta t} \frac{\rho}{t} (R_{c\theta t} - \tilde{R}_{c\theta t})(1 - F_{\theta t}). \]  

(2.48)

In the free-stream the transported value of the transition momentum thickness Reynolds number \( \tilde{R}_{c\theta t} \) is attracted to the value calculated using the empirical correlation \( R_{c\theta t} \), while in the boundary layer this term vanishes to allow the transported value of \( \tilde{R}_{c\theta t} \) to convect and diffuse into the boundary layer. This process is controlled by the blending function, \( F_{\theta t} \), which is designed to switch rapidly from a value of zero in the free-stream to unity in the boundary layer. This function is defined as follows:

\[ F_{\theta t} = \min \left( \max \left( F_{\text{wake}} e^{-\left( \frac{\phi}{\delta} \right)^4}, 1 - \left( \frac{\gamma - 1/c_{e2}}{1 - 1/c_{e2}} \right)^2 \right), 1 \right), \]  

(2.49)

\[ \delta = \frac{50 Sd}{d_{BL}}, \]  

(2.50)

\[ \delta_{BL} = \frac{15}{2} \theta_{BL}, \]  

(2.51)

\[ \theta_{BL} = \frac{\tilde{R}_{c\theta t} \mu}{\rho U}. \]  

(2.52)

The wake function \( F_{\text{wake}} \), which is used to disable the blending function in the wake regions downstream of the airfoil, is defined in the work produced by Schucker [47] as

\[ F_{\text{wake}} = e^{-\frac{R_{cU_{i,j}}}{500/\mu}}, \]  

(2.53)

where

\[ R_{cU_{i,j}} = \frac{\rho d^2}{\mu} \left( \frac{\partial U_i}{\partial x_j} \frac{\partial U_j}{\partial x_i} \right). \]  

(2.54)

The variable \( t \) is a time scale present in \( P_{\theta t} \) for dimensional reasons given by *

\[ t = \frac{500 \mu}{\rho U^2}. \]  

(2.55)

The constant \( c_{\theta t} \) controls the magnitude of the production term, while the constant \( \sigma_{\theta t} \) controls the diffusion process. The value of these constants are as follows:

\[ c_{\theta t} = 0.03, \]  

\[ \sigma_{\theta t} = 2.0. \]

**Seperation-Induced Transition**

Langtry noted that in the presence of laminar boundary layer separation the transition model would predict the location of flow reattachment further downstream of the location given by experiment [21]. To correct this issue, a modification to the model was developed which allows intermittency to grow to values greater than one wherever laminar separation is detected. This would lead to an increase in the production of turbulence in the boundary layer, resulting in reattachment of the flow. It is important

*The variable \( t \) used here should not be confused with the time variable \( t \) used elsewhere in this thesis.*
to note that in the $\gamma - Re_{\theta_t}$ model implementation $\gamma$ is treated as a model parameter and not a physical value for intermittency; therefore a value of intermittency greater than one is acceptable.

The modification is defined as

$$\gamma_{sep} = \min\left(s_1 \max\left(0, \left(\frac{Re_{\nu}}{3.235 Re_{\theta c}}\right) - 1\right) F_{reattach}, 2\right) F_{\theta t},$$

(2.56)

$$s_1 = 2$$

(2.57)

The growth of the $\gamma_{sep}$ value is triggered when the correlation,

$$\frac{Re_{\nu}}{3.235 Re_{\theta c}},$$

(2.58)

exceeds a value of 1. This is similar to the transition criterion used to trigger the production of intermittency in equation (2.42); however, its scaling has been modified to account for a free shear layer [21].

The constant $s_1$ controls the size of the separation bubble. The reattach function is designed to disable the modification when reattachment is detected, and is defined as

$$F_{reattach} = e^{-(\frac{Re_{\nu}}{3.235})^4}.$$  

(2.59)

The blending function, $F_{\theta t}$, has been included to ensure the modification is only active in the boundary layer. Furthermore, the modification is designed to have a negligible effect on the predictions for attached boundary-layer transition through the use of the function given by,

$$\gamma_{eff} = \max(\gamma, \gamma_{sep}).$$

(2.60)

where the $\gamma_{sep}$ value is only used when separation, and therefore a value of intermittency greater than 1, is detected.

**Coupling to Spalart-Allmaras turbulence model**

The $\gamma - Re_{\theta_t}$ transition model is coupled to the Spalart-Allmaras turbulence model [52] through the modification of the production term, which is defined as

$$\tilde{P}_{\nu} = \gamma_{eff} P_{\nu},$$

(2.61)

where $P_{\nu}$ is the original production term in the Spalart-Allmaras model, represented in equation (2.10), and $\gamma_{eff}$ is the effective value for intermittency, equation (2.62). In their implementation, Medida and Baeder [30] proposed a modification to the destruction term given by,

$$\tilde{D}_{\nu} = \min(\max(\gamma, \beta), 1) D_{\nu},$$

(2.62)

where $D_{\nu}$ is the original destruction term, and $\beta$ controls the magnitude of the destruction term. However, it was shown in the work performed by Shucker [47] that assuming a constant $\beta$ value of 1, resulting in no modification of the destruction term, had little effect on the turbulent and transitional regions. Therefore, the destruction term will not be modified in the current work.
2.3.2 Curvilinear Coordinate Transformation

For the simulations used in the current work, the physical space around a solid body is represented by discrete points within a structured grid. To simplify the process of discretizing the governing equations to this grid, a curvilinear coordinate transformation is applied to the equations to transform them from physical coordinates \((x, y, z)\) to a computational coordinate system \((\xi, \eta, \zeta)\). The coordinate transformation is defined as

\[
\begin{align*}
\xi &= \xi(x, y, z), \\
\eta &= \eta(x, y, z), \\
\zeta &= \zeta(x, y, z),
\end{align*}
\]

while grid spacings for the computational grid are specified as

\[
\Delta \xi = \Delta \eta = \Delta \zeta = 1.
\]

The curvilinear coordinate transformation of the Navier-Stokes and Spalart-Allmaras turbulence model equations are provided in the work of Osusky and Zingg [36]. The transformed transition model equations, equations (2.35) and (2.48), are given by

\[
\partial_t (J^{-1} \rho \gamma) + \partial_\xi (J^{-1} \rho U \gamma) + \partial_\eta (J^{-1} \rho V \gamma) + \partial_\zeta (J^{-1} \rho W \gamma) = \frac{1}{Re} \left[ \frac{\partial}{\partial \xi} \left( J^{-1} \left( \mu_t + \frac{\mu_t}{\sigma_f} \right) \left( \xi_x \frac{\partial \gamma}{\partial x} + \xi_y \frac{\partial \gamma}{\partial y} + \xi_z \frac{\partial \gamma}{\partial z} \right) \right) + \frac{\partial}{\partial \eta} \left( J^{-1} \left( \mu_t + \frac{\mu_t}{\sigma_f} \right) \left( \eta_x \frac{\partial \gamma}{\partial x} + \eta_y \frac{\partial \gamma}{\partial y} + \eta_z \frac{\partial \gamma}{\partial z} \right) \right) + \frac{\partial}{\partial \zeta} \left( J^{-1} \left( \mu_t + \frac{\mu_t}{\sigma_f} \right) \left( \zeta_x \frac{\partial \gamma}{\partial x} + \zeta_y \frac{\partial \gamma}{\partial y} + \zeta_z \frac{\partial \gamma}{\partial z} \right) \right) \right]
\]

where

\[
\begin{align*}
\frac{\partial \gamma}{\partial x} &= \xi_x \partial_\xi \gamma + \eta_x \partial_\eta \gamma + \zeta_x \partial_\zeta \gamma \\
\frac{\partial \gamma}{\partial y} &= \xi_y \partial_\xi \gamma + \eta_y \partial_\eta \gamma + \zeta_y \partial_\zeta \gamma \\
\frac{\partial \gamma}{\partial z} &= \xi_z \partial_\xi \gamma + \eta_z \partial_\eta \gamma + \zeta_z \partial_\zeta \gamma
\end{align*}
\]

and

\[
\partial_t (J^{-1} \rho \theta t) + \partial_\xi (J^{-1} \rho U \theta t) + \partial_\eta (J^{-1} \rho V \theta t) + \partial_\zeta (J^{-1} \rho W \theta t) = \frac{1}{Re} \left[ \frac{\partial}{\partial \xi} \left( J^{-1} \left( \sigma_{\theta t} + \mu_t \right) \left( \xi_x \frac{\partial \theta t}{\partial x} + \xi_y \frac{\partial \theta t}{\partial y} + \xi_z \frac{\partial \theta t}{\partial z} \right) \right) + \frac{\partial}{\partial \eta} \left( J^{-1} \left( \sigma_{\theta t} + \mu_t \right) \left( \eta_x \frac{\partial \theta t}{\partial x} + \eta_y \frac{\partial \theta t}{\partial y} + \eta_z \frac{\partial \theta t}{\partial z} \right) \right) + \frac{\partial}{\partial \zeta} \left( J^{-1} \left( \sigma_{\theta t} + \mu_t \right) \left( \zeta_x \frac{\partial \theta t}{\partial x} + \zeta_y \frac{\partial \theta t}{\partial y} + \zeta_z \frac{\partial \theta t}{\partial z} \right) \right) \right]
\]
where

\[
\begin{align*}
\frac{\partial \tilde{Re}_{\theta l}}{\partial x} &= \xi_x \partial_x \tilde{Re}_{\theta l} + \eta_x \partial_y \tilde{Re}_{\theta l} + \zeta_x \partial_z \tilde{Re}_{\theta l} \\
\frac{\partial \tilde{Re}_{\theta l}}{\partial y} &= \xi_y \partial_x \tilde{Re}_{\theta l} + \eta_y \partial_y \tilde{Re}_{\theta l} + \zeta_y \partial_z \tilde{Re}_{\theta l} \\
\frac{\partial \tilde{Re}_{\theta l}}{\partial z} &= \xi_z \partial_x \tilde{Re}_{\theta l} + \eta_z \partial_y \tilde{Re}_{\theta l} + \zeta_z \partial_z \tilde{Re}_{\theta l}.
\end{align*}
\]

The metric Jacobian inverse is defined as $J^{-1}$, while $(U, V, W)$ denote the contravariant velocity components.

### 2.3.3 Initial Conditions

The transition momentum thickness Reynolds number is initialized with the value from a zero pressure gradient calculation of the empirical correlation using the experimental turbulence intensity, $Tu$. For cases where an experimental turbulence intensity is not specified, this value must be approximated to represent free-stream conditions. Intermittency is specified as a value of unity everywhere in the flow domain. Conceptually, this differs from the physical definition of intermittency where $\gamma$ is equal to zero in the free-stream and given a value of unity in turbulent boundary layers; however, this method, first implemented in the intermittency transport equation by Steelant and Dick [53], has been shown to increase the performance of the transition model in regions of stagnated flow [21].

### 2.3.4 Boundary Conditions

**Farfield Boundary**

The boundary condition for $\tilde{Re}_{\theta l}$ in the farfield is the zero pressure gradient solution of the empirical correlation, while intermittency is given a value of unity in the farfield.

**Solid Surface Boundary**

At a solid surface the boundary conditions for $\tilde{Re}_{\theta l}$ and $\gamma$ are zero flux normal to the wall, defined as

\[
\begin{align*}
\frac{\partial \tilde{Re}_{\theta l}}{\partial n} &= 0, \\
\frac{\partial \gamma}{\partial n} &= 0.
\end{align*}
\]

**Symmetry Boundary**

As with a solid surface boundary, the boundary condition for $\tilde{Re}_{\theta l}$ and $\gamma$ at a symmetry boundary is zero flux.
Chapter 3

Spatial Discretization

The governing equations presented in Chapter 2 are spatially discretized using Summation-by-Parts (SBP) operators, with Simultaneous Approximation Terms (SATs) applied to enforce boundary conditions and inter-block coupling. The following chapter discusses the grid generation process, the SBP operators for the first and second derivatives, and the SATs for various boundaries. In addition, the application of the SBP operators to the transition model equations is presented. The implementation of the SBP operators for the Euler and Navier-Stokes equations is discussed in Hicken and Zingg [13] and Osusky and Zingg [36], respectively.

3.1 Domain Decomposition

The computational domain is decomposed into multiple blocks, which results in multi-block structured grids that are ideal for use with SBP-SAT discretization. This allows for parallel computation of the domain, reducing the computational time required for each simulation. In addition, this type of blocking strategy greatly simplifies the creation of meshes around complex geometries, where the grid density can be increased in sensitive areas without affecting the grid resolution elsewhere [35, 36]. Moreover, several different types of grid topologies can be used depending on the shape of the aerodynamic body. As an example, Figure 3.1 demonstrates suitable grid blocking for a NACA0012 airfoil with a sharp trailing edge.

3.2 Summation-by-Parts Operators

Finite-difference approximations are used in the current work to approximate the first and second derivatives found in the governing equations. These are applied in the form of SBP operators, which are second-order centered difference schemes that do not include information about the boundaries; this information is supplied using SATs. SBP operators were originally derived by Kreiss and Scherer [15], extended by Strand [55], and have been shown to be time-stable for the linearized Navier-Stokes equations [26]. However, for curvilinear coordinates, SBP operators must be constructed with a diagonal norm matrix to ensure time-stability [35]. First and second derivatives appear in the $\gamma - Re_{\theta t}$ transition model as $\partial_\xi \alpha$ and $\partial_\xi (\beta) \partial_\xi \alpha$, respectively, where $\beta$ is a spatially varying coefficient and $\alpha$ here refers to an arbitrary flow quantity and not the angle of attack.
First Derivative

The SBP operator for the first derivative is given by

$$D_1 = H^{-1} \Theta,$$

(3.1)

where

$$\Theta = \frac{1}{2} \begin{bmatrix}
-1 & 1 & & \\
-1 & 0 & 1 & \\
& \ddots & \ddots & \ddots \\
& & -1 & 0 & 1 \\
& & & -1 & 1
\end{bmatrix}.$$

In the interior of the flow the operator is a second-order centered difference approximation, while at the boundaries it converts to a one-sided first-order approximation. The diagonal norm matrix $H$, which is required to guarantee time-stability, is specified as

$$H = h \begin{bmatrix}
\frac{1}{2} & & \\
& 1 & & \\
& & \ddots & \\
& & & 1 \\
& & & \frac{1}{2}
\end{bmatrix}.$$

The value of the mesh spacing, represented by $h$, is equal to unity due to the uniform computational grid, where $\Delta \xi$, $\Delta \eta$, and $\Delta \zeta$ are all equal to unity.
Second Derivative

To approximate second derivatives, the compact SBP operator developed by Mattson et al. [26] is applied. An alternative approach is to apply the first derivative SBP operator twice to approximate a second derivative; however, this approach has been shown to increase global error, and is less effective at dissipating high wavenumber modes, while reducing the order of accuracy by one [35].

The compact second-order SBP operator for a second derivative is given by

\[
D_2(\beta) = H^{-1}\{ - (D_1)^T H B D_1 - \frac{1}{4h} (\tilde{D}_2)^T C_1 B \tilde{D}_2 + EBD_1^{(2)} \},
\]

(3.2)

where

\[
\tilde{D}_2 = \begin{bmatrix}
1 & -2 & 1 \\
1 & -2 & 1 \\
\ddots & \ddots & \ddots \\
1 & -2 & 1 \\
1 & -2 & 1
\end{bmatrix}, \quad C_1 = \begin{bmatrix}
0 & & \\
1 & & \\
& \ddots & \\
& 1 & \\
& & 0
\end{bmatrix},
\]

and

\[
EBD_1^{(2)} = \frac{1}{h} \begin{bmatrix}
\frac{3\beta_1}{2} & -2\beta_1 & \frac{\beta_1}{2} \\
0 & 0 & 0 \\
\ddots & \ddots & \ddots \\
0 & 0 & 0 \\
\frac{\beta_N}{2} & -2\beta_N & \frac{3\beta_N}{2}
\end{bmatrix}.
\]

B is a diagonal matrix containing the spatially varying coefficients, while at block boundaries the notation \(D^{(b)}\) represents the order of the operator, and the notation ‘˜’ signifies an undivided difference operator.

Application to Transition Model Equations

First derivatives appear in the advective terms for the transition model equations, equations (2.64) and (2.65). Specifically, they appear in the form

\[
\partial_\xi (U\alpha),
\]

(3.3)

where \(U\) is a contravariant velocity component, and the spatial derivative is being taken in the \(\xi\) direction. The first-derivative SBP operators can be applied to these terms as

\[
\partial_\xi (U\alpha) \approx D_1\xi U\alpha,
\]

(3.4)

where \(\alpha\) is a vector containing the flow quantities, \(\alpha\), and \(U\) is a diagonal matrix containing the contravariant velocities, \(U\). In addition, these operators can be applied to the cross derivatives, which appear in the diffusive terms, as

\[
\partial_\xi (\beta \partial_\eta \alpha) \approx D_1\xi \beta D_1\eta \alpha,
\]

(3.5)
where $\beta$ is a diagonal matrix containing the spatially varying coefficients. The double derivatives in the diffusive terms are approximated using the second-derivative SBP operators as

$$
\partial_\xi (\beta \partial_\xi \alpha) \approx D_{2\xi} \beta \alpha.
$$

(3.6)

Numerical dissipation is added to the discretization of the transition and turbulence model equations to produce a first-order upwinding operator from the second-order centered difference operator. Using the formulation presented by Osusky [35], the first-derivative operator can be expressed as a dissipation operator added to the centered difference operator, given by

$$
\partial_\xi (U \alpha) \approx D_{1\xi} U \alpha + \frac{1}{2} |U| H^{-1} D_d^T C_1 D_d \alpha,
$$

(3.7)

where

$$
D_d = \begin{bmatrix}
-1 & 1 \\
-1 & 1 \\
\vdots & \ddots \\
-1 & 1 \\
1 & 1
\end{bmatrix}, \quad C_1 = \begin{bmatrix}
1 \\
\vdots \\
1 \\
0
\end{bmatrix}.
$$

3.3 Simultaneous Approximation Terms

SATs are used to enforce boundary conditions and inter-block coupling. The SBP-SAT approach requires less information from adjoining blocks than alternative methods, such as the halo-node approach [36]. For domain boundaries the node information required from the adjacent block is replaced with a boundary condition. For parallel algorithms, large amounts of information sharing between blocks can result in excessive inter-processor communication times. The reduction in node information required from adjoining blocks with the SBP-SAT approach allows for less inter-processor communication and information sharing, resulting in a more efficient algorithm. This effect is amplified with the implementation of higher-order discretizations, which require large computational stencils. In addition, the continuity requirements for mesh generation at surfaces are reduced for the SBP-SAT approach, as it does not involve taking derivatives across the block interface. Only $C^0$ continuity is required, which allows for accurate solutions to be obtained on grids with high angles of grid line incidence at interfaces.

The focus of this section is to present the application of the various SATs to the transition model equations. The implementation of these SATS with the Navier-Stokes equations and SA turbulence model was provided by Osusky [35]. The SAT terms developed for the transition model follow the methodology used for the turbulence model equations. The SATs for the advective portions of the transition model equations are defined as

$$
SAT_{\text{adv}} = J^{-1} H_b^{-1} \sigma_a (\alpha - \alpha_{\text{target}}),
$$

(3.8)

where $\alpha$ represents the local value of the transition model variables, $\gamma$ and $Re_{\theta t}$, and $\alpha_{\text{target}}$ is the target value of the transition model variables. This is specified by a boundary condition at a domain boundary, or the corresponding value on an adjoining block at a block interface. $H_b$ is the boundary node element of the diagonal norm matrix $H$, and $J$ is the metric Jacobian. The parameter $\sigma_a$ provides the direction.
Chapter 3. Spatial Discretization

27

of information propagation in the flow, and is defined as

\[ \sigma_a = -\frac{1}{2} \max([|U|, \phi] + \delta_a U). \] (3.9)

where \( \delta_a \) is given a value of \(-1\) on the high side of a block, and a value of \(+1\) on the low side. To prevent
the SAT from disappearing in areas where the contravariant component of velocity is zero the limiting
factor \( \phi \) is included, which is given by

\[ \phi = V_l \left( |U| + \alpha \sqrt{\xi_x^2 + \xi_y^2 + \xi_z^2} \right), \] (3.10)

where the constant \( V_l \) is given a value of 0.025, and \( \alpha \) represents the speed of sound. At an interface, the
flow variables in \( \sigma_a \), such as the contravariant velocity, sound speed, and density, are calculated based
on an average of the two sides of the interface, while at a domain boundary they are constructed using
only local information from the interior of the flow. The grid metrics are determined using the local
block information.

The SATs for the diffusive portions of the transition model equations are divided into two parts, the
first dealing with the difference in the transition model quantity gradient, and the second dealing with
the difference in the transition model quantity. The SAT dealing with the transition model quantity
gradient is specified as

\[ SAT_{qnty} = -H_b^{-1} \sigma_{qnty} (\alpha - \alpha_{target}). \] (3.11)

where \( \sigma_{qnty} \) is given a value of \(-1\) on the high side of a block and a value of \(+1\) on the low side. The
gradients for the transition model quantities, represented by \( g \), are given by

\[ g = \frac{1}{Re} J^{-1} \left( \mu + \mu_t \sigma_f \right) \left[ (\xi_x^2 + \xi_y^2 + \xi_z^2) \partial_t \gamma + (\xi_x \xi_y + \xi_y \xi_z + \xi_z \xi_x) \partial_t \eta \gamma + (\xi_x \xi_z + \xi_y \xi_y + \xi_z \xi_z) \partial_t \zeta \gamma \right], \] (3.12)

\[ g = \frac{1}{Re} J^{-1} \sigma_{\theta t} (\mu + \mu_t) \left[ (\xi_x^2 + \xi_y^2 + \xi_z^2) \partial_t \Re_{e \theta t} + (\xi_x \xi_y + \xi_y \xi_z + \xi_z \xi_x) \partial_t \Re_{\eta \theta t} \right. \]

\[ + (\xi_x \xi_z + \xi_y \xi_y + \xi_z \xi_z) \partial_t \Re_{\zeta \theta t}, \] (3.13)

for the intermittency and transition momentum thickness Reynolds number equations, respectively. The constants \( \sigma_f \) and \( \sigma_{\theta t} \) are part of the transition model equation definition with values of 1.0 and 2.0. At
a farfield boundary, the target flux for each equation is given a value of zero, while at a block interface
the target gradient is calculated based on the value of the flow variables at the interface of the adjoining
block.

The diffusive SAT dealing with the transition model quantity is specified as

\[ SAT_{flux} = -H_b^{-1} \sigma_{flux} (g - g_{target}). \] (3.14)

where \( \sigma_{flux} \) for the intermittency and transition momentum thickness Reynolds number equation is given
by

\[ \sigma_{flux} = J^{-1} \left( \mu + \mu_t \right) \left[ (\xi_x^2 + \xi_y^2 + \xi_z^2) \right], \] (3.15)

\[ \sigma_{flux} = J^{-1} \sigma_{\theta t} (\mu + \mu_t) \left[ (\xi_x^2 + \xi_y^2 + \xi_z^2) \right], \] (3.16)
respectively. As with the advective SAT, at a block interface the flow variables in $\sigma_{dv}$ are calculated based on an average of the two sides of the interface, while at a domain boundary they are constructed using local information from the interior of the flow.
Chapter 4

Solution Algorithm

The application of the SBP-SAT spatial discretization, discussed in Chapter 3, to the Navier-Stokes equations, SA turbulence model, and $\gamma - \bar{Re}_\theta$ transition model equations, leads to a nonlinear system of ordinary differential equations of the form

$$\frac{dQ}{dt} + R(Q) = 0,$$  \hspace{1cm} (4.1)

where $Q$ is a vector of the conserved, Spalart-Allmaras, and transition model variables at each node in the mesh, and $R$ represents the residual vector. The objective of this thesis is to investigate the performance of the $\gamma - \bar{Re}_\theta$ transition model for steady flows; therefore equation (4.1) can be simplified to the nonlinear algebraic system of equations given by

$$R(Q) = 0.$$ \hspace{1cm} (4.2)

The nonlinear system of equations is driven from an initial state to a converged steady-state solution using an iterative strategy. The solution strategy applied for the current algorithm is discussed in this chapter.

4.1 Newton-Krylov Approach

Newton’s method is a widely adopted nonlinear solver, which will converge quadratically if an acceptable initial guess is supplied. Based on previous work performed in our group [13, 36], a pseudo-transient continuation (PTC) strategy is applied in order to determine this initial iterate. This globalization strategy is similar to an implicit Euler time-marching discretization strategy, providing the algorithm with increasing time step values during the solution process, but with several modifications, which will be discussed in the following sections. An implicit time marching strategy was selected as the solution strategy for the present algorithm for several reasons. For RANS simulations, high-aspect-ratio cells are generated in the boundary layer of the aerodynamic body, which cause slow convergence of explicit methods due to a severe time-step restriction. An implicit strategy allows for large time steps, and when combined with a local time linearization, can provide stable and efficient convergence.

Applying the implicit Euler time marching method with local time linearization to the discrete equations results in a large system of linear equations given by
Chapter 4. Solution Algorithm

\[(T^{(n)} + A^{(n)})\Delta Q^{(n)} = -R(Q^{(n)}), \tag{4.3}\]

where \(T\) is a diagonal matrix containing the inverse local time step values, \(n\) represents the nonlinear iteration index, and

\[\Delta Q^{(n)} = Q^{(n+1)} - Q^{(n)}.\]

The flow Jacobian \(A\) is defined as

\[A^{(n)} = \frac{\partial R^{(n)}}{\partial Q^{(n)}}. \tag{4.4}\]

As the local time step values approach infinity, the implicit Euler method evolves into Newton’s method, given by

\[(A^{(n)})\Delta Q^{(n)} = -R(Q^{(n)}), \tag{4.5}\]

The strategy consists of two phases, the approximate-Newton and inexact-Newton. The approximate-Newton phase determines a suitable initial iterate for the inexact-Newton method by converging the residual by several orders of magnitude, while the inexact-Newton phase converges the system to a steady-state flow solution. GMRES [40], a Krylov iterative solver, is used with a parallel preconditioner to solve the large system of linear equations generated at each iteration.

The following sections describe the approximate and inexact-Newton phases, and the process of switching between these phases.

4.1.1 Approximate-Newton Phase

The approximate-Newton phase is applied to determine a suitable initial iterate for the inexact-Newton method. Since we are interested in an initial guess and not a time-accurate solution, several modifications can be applied to the implicit Euler time marching method to improve convergence. These include the implementation of a first-order, approximate flow Jacobian, a lagged Jacobian update, and local time stepping. The implementation of an approximate Jacobian has been shown to be effective during the start-up phase [5, 34, 38].

There are several methods for constructing the approximate flow Jacobian, which represents an approximation to the partial derivative of the flow residual with respect to the conserved variables. These include finite-difference and complex step methods, automatic differentiation, and analytical differentiation. Although the most challenging to implement, analytical differentiation was used for the majority of the linearization of the flow equations due to its fast execution time, with the complex step and finite-difference methods being used for select entries. To generate a first-order Jacobian, cross-derivatives that appear in the viscous terms are not included in the linearization. In addition, only information from nearest neighbour nodes are used, while tangential derivatives in the formation of the SATs for the governing equations are neglected. Further information regarding the formation of the approximate flow Jacobian can be found in the following references [13, 35]. The linearization of the transition model equations for the first-order Jacobian follows the Spalart-Allmaras turbulence model equation linearization, with the cross-derivative terms being neglected.
The preconditioner for GMRES is produced using an incomplete lower-upper factorization of the first-order Jacobian with a fill level of 3, which was demonstrated in the work by Osusky [35] to produce the fastest overall convergence. This factorization is computationally expensive, and if completed at each nonlinear approximate-Newton iteration can result in excessive computation time. To increase efficiency, the method suggested by Kim and Orkwis [14] of updating and factoring the Jacobian periodically, rather than at each outer iteration, has been adopted. Updating the preconditioner at iteration \( n \) if \( \text{mod}(n, m) = 0 \), where \( m = (3, 4, 5) \), has been shown to provide a good compromise between CPU time and robustness [13]; however, for more complex flows the value of \( m \) must be reduced [35].

For Newton-Krylov algorithms the use of a spatially varying time step has been shown to improve the rate of convergence [35]. The local time step is specified as

\[
\Delta t^{(n)}_{j,k,m} = \frac{J_{j,k,m} \Delta t^{(n)}_{\text{ref}}}{1 + \sqrt{J_{j,k,m}}},
\]

for three-dimensional flows and

\[
\Delta t^{(n)}_{j,k} = \frac{J_{j,k} \Delta t^{(n)}_{\text{ref}}}{1 + \sqrt{J_{j,k}}},
\]

for two-dimensional flows, where the reference time step is defined as

\[
\Delta t^{(n)}_{\text{ref}} = a(b)^n.
\]

Stable values of the reference time step for fully turbulent and fixed transition flows were found in the work conducted by Osusky and Zingg to be \( a = 0.001 \) and \( b \in [1.05, 1.5] \) [35]. The optimal values of the time step variables for flows with transition, which require additional stability measures, are discussed in Chapter 5. The metric Jacobian \( J \) is included in the numerator, since the unscaled flow variables \( Q \) are used by the solver rather than the transformed variables \( \tilde{Q} \).

### 4.1.2 Inexact-Newton Phase

The inexact-Newton phase is designed to accelerate the convergence of the solution once the approximate-Newton phase produces a suitable initial iterate. As the reference time step approaches infinity, the inverse time step matrix from the left-hand-side of the discretized governing equations disappears, which transforms the implicit Euler time-marching method into Newton’s method. Thus accelerated convergence is achieved through a more aggressive reference time step, which is based on the successive evolution relaxation method developed by Mulder and van Leer [33], and is given by

\[
\Delta t^{(n)}_{\text{ref}} = \max\left[\alpha\left(R_d^{(n)}\right)^{-\beta}, \Delta t^{(n-1)}_{\text{ref}}\right].
\]

For the present work the constant \( \beta \) is given a value of 2. The max function is included to ensure that the reference time step does not decrease. This is useful during simulations where the residual norm may increase temporarily during convergence. The transition between the approximate and inexact-Newton reference time steps is controlled using the function \( \alpha \), which is defined as

\[
\alpha = a(b)^{n_{\text{Newt}}} \left(R_d^{(n_{\text{Newt}})}\right)^{\beta},
\]
where the variable $n_{\text{Newt}}$ represents the first inexact-Newton iteration, and the reference time step values, $a$ and $b$, are the same as those used in equation (4.8). The function is designed to ensure a smooth transition between the two phases, avoiding an abrupt increase or decrease in the value of the reference time step. The value $R_d^{(n)}$ represents the relative drop in the residual, which is specified as

$$R_d^{(n)} = \frac{||R^{(n)}||_2}{||R^{(0)}||_2},$$

where $R^{(0)}$ is the residual of the first iteration of the approximate Newton phase.

The inexact-Newton phase differs from the approximate Newton phase in regard to the Jacobian as well. As opposed to the first-order approximate Jacobian used in the approximate Newton phase, a full, second-order accurate Jacobian, defined in equation (4.4), is used for the inexact-Newton phase. However, since the Krylov subspace method is used to solve the linear system, equation (4.3), the full flow Jacobian can be represented with Jacobian-vector products. To simplify the calculation of these products and reduce memory requirements, a first-order forward difference approximation of the Frechet derivative is used to approximate these terms:

$$A^{(n)} v \approx \frac{R(Q^{(n)} + \epsilon v) - R(Q^{(n)})}{\epsilon},$$

where the parameter $\epsilon$ is given by

$$\epsilon = \sqrt{N_u \delta v^T v}.$$ 

The value $N_u$ represents the number of unknowns, while $\delta$ is a perturbation parameter. The value of the perturbation parameter influences the magnitude of truncation errors and round-off errors [35], and therefore should be chosen carefully. For this work a value of $\delta = 10^{-12}$ is used. Although the full Jacobian matrix is not required to be computed or stored, due to the use of Frechet derivatives to approximate the Jacobian-vector products, it is still necessary to compute and store the first-order approximate Jacobian, $A_1$, which is used to precondition the linear system. For both the approximate and inexact-Newton phases it is not necessary to solve the linear system exactly. The relative tolerance to which the linear system is solved is given by

$$||R^{(n)} + A^{(n)} \Delta Q^{(n)}||_2 \leq \eta_n ||R^{(n)}||_2,$$

where the forcing parameter $\eta_n$ controls the level to which the linear system is converged. Over-solving the linear system will increase the computational cost of the linear solver, while under-solving the system will affect the convergence of the nonlinear solver. For the current work $\eta_n$ is given a value of 0.05 for the approximate Newton phase, and a value of 0.01 for the inexact-Newton phase. These values were determined by Osusky and Zingg [35] to produce efficient convergence for viscous RANS simulations, and perform well with fully turbulent flows and flows with laminar-turbulent transition.

### 4.1.3 Switching Between Phases

The transition from the approximate-Newton phase to the inexact-Newton phase is an important part of the PTC strategy. The switch between the phases is initiated when the value of the nonlinear residual, $R_d^{(n)}$, drops below a threshold value, $\tau$. The value of this parameter is critical to the convergence of
the solver. A choice of \( \tau \) that is too small can cause the approximate Newton phase to be inefficient or possibly stall, while a large \( \tau \) value can cause the solver to switch to the inexact-Newton phase before a sufficient initial guess has been found, leading to possible divergence of the solution.

In the start-up phase, for flows simulating transition, the residual oscillates for several iterations before a convergence pattern is evident. For these reasons the relative residual drop threshold was adjusted from that used for laminar or turbulent flows to compensate for these differences. A value for \( \tau \) of \( 10^{-6} \) was found to be optimal. The choice of this value is discussed in Chapter 5.

### 4.2 Preconditioning

The efficiency of the linear solver is dependent on the accuracy of the preconditioner. The approximate-Schur parallel preconditioner used is based on an incomplete lower-upper factorization with a fill level of 3 of the local sub matrix of the modified Jacobian, given by [13]:

\[
L_i U_i = \left[ T^{(n)} + A_1^{(n)} \right] + R_i, \tag{4.15}
\]

The application of this preconditioner to the Newton-Krylov flow solver is discussed in references [13, 35].

### 4.3 Equation and Variable Scaling

A scaling strategy has been implemented for the current work in order to obtain an efficient and accurate solution of the linear system. It is based on the work by Chisholm and Zingg [7], which was further developed by Osusky and Zingg [35, 36]. Instead of solving the linear system of equations directly, the linear solution algorithm solves a scaled version of the equations given by

\[
S_a S_r \left( \frac{I}{\Delta t} + A^{(n)} \right) S_c S_c^{-1} \Delta Q^{(n)} = -S_a S_r R^{(n)}, \tag{4.16}
\]

where \( S_r \) and \( S_c \) represent the row and column scaling matrices, which scale the system equations and variables, respectively. The individual equation components are brought to within an order of magnitude of each other using the auto-scaling matrix \( S_a \). The entries in the auto-scaling matrix are calculated using the equation-wise residual \( L_2 \)-norms of the partially scaled system \( S_r R^{(n)} \).

The row and column matrices are specified as

\[
S_r = \text{diag}(S_{r1}, ..., S_{rN}), \quad S_c = \text{diag}(S_{c1}, ..., S_{cN}), \tag{4.17}
\]

where
The Spalart-Allmaras turbulence model equation and the $\tilde{\text{Re}}_{\theta_t}$ equation from the transition model are scaled by the respective maximum values, $\tilde{\nu}_{\text{max}}$ and $\tilde{\text{Re}}_{\theta_{t\text{max}}}$ in the flow domain. The large variable quantities produced by these equations can destabilize the linear solution, leading to inaccurate results and unpredictable behaviour of the linear and nonlinear solvers. This scaling ensures the variables are close in magnitude to the mean flow equations, effectively normalizing the turbulence and $\tilde{\text{Re}}_{\theta_t}$ equations. The values of these terms affect the numerical properties of the linear system, especially the incomplete lower-upper factorization of the Jacobian and the Frechet derivative. Osusky and Zingg [35] demonstrated that replacing the $\tilde{\nu}_{\text{max}}$ term with a constant value of $10^3$ resulted in accurate and efficient convergence of the linear system. Simulations have shown that the $\tilde{\text{Re}}_{\theta_t}$ equation produces similar maximum values as the SA turbulence model. Therefore, to reduce the computational cost associated with conducting a global search of the domain for these maximum values, in the current work $\tilde{\nu}_{\text{max}}$ and $\tilde{\text{Re}}_{\theta_{t\text{max}}}$ are given constant values of $10^3$. The intermittency equation produces variables in the range between 0 and 1, which is similar to the mean flow equations, and therefore does not require additional equation scaling measures.

The Navier-Stokes equations have been scaled by the local value of the metric Jacobian, $J_i$. This is to account for the inverse metric Jacobian terms present in the equations from the coordinate transformation, discussed in Section 2.3.1 [13, 35]. The metric Jacobian differs in value greatly in different regions of the flow. For example, large values of $J$ are present in the nodes close to the surface, while smaller values are found in the free-stream. These large differences in the scales of the linear system entries negatively affect the performance of the linear solver if left unaccounted for. The turbulence model and transition model equations do not contain this inherent geometric scaling, and therefore the metric Jacobian is not included in their scaling.

It is important to note that for the calculation of the reference time step in the inexact-Newton
phase, given by equation (4.9), the relative drop in the residual, $R_d^{(n)}$, is calculated using the partially scaled residual, $S, R^{(n)}$. 

Chapter 5

Algorithm Optimization

The transition model equations are a highly nonlinear set of equations, which can negatively affect stability and convergence. In order to improve the robustness of the solution algorithm, modifications were made to the nonlinear flow solver and the performance of the negative Spalart-Allmaras model was examined.

The goal of this chapter is to provide parameters and introduce modifications to the algorithm, which will result in an efficient, robust, and accurate flow solution algorithm. A set of test cases was used in the optimization of the current algorithm, which is discussed in the following section.

5.1 Test Cases

The geometries and grids used for the optimization of the algorithm are listed in Table 5.1, while the flow characteristics used for the test cases are listed in Table 5.2.

The focus of the simulations in this work is on subsonic, transitional flow, which represent the flow conditions used by Langtry and Menter [21] in the development of the original $\gamma - \tilde{Re}_{\theta_t}$ transition model. Low Mach numbers and angles of attack were chosen to be representative of these flows. The turbulence intensity was varied to simulate different levels of free-stream turbulence. These values were selected from the flat plate experiments, and represent the upper and lower bound of turbulence intensity investigated in the studies [45, 46]. The empirical correlations in the $\gamma - \tilde{Re}_{\theta_t}$ transition model are based on two-dimensional boundary layer theory. To reduce the computational cost of these test cases and allow a larger number of tests to be conducted two-dimensional grids and geometries were used in the optimization. Specifically the geometries used for the optimization consist of a flat plate test case, and a NACA0012 airfoil. It is important to note that the parameters and modifications discussed in this chapter were also used to produce the results presented in Chapter 6. The grid topologies used for these

<table>
<thead>
<tr>
<th></th>
<th>Geometry</th>
<th>Topology</th>
<th>Blocks</th>
<th>Nodes</th>
<th>Off-wall spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>flat plate</td>
<td>H</td>
<td>1</td>
<td>13,289</td>
<td>$2.0 \times 10^{-6}$</td>
</tr>
<tr>
<td>2</td>
<td>NACA0012 airfoil</td>
<td>C-grid</td>
<td>32</td>
<td>60,320</td>
<td>$1.69 \times 10^{-6}$</td>
</tr>
</tbody>
</table>
Table 5.2: Flow conditions for test cases

<table>
<thead>
<tr>
<th></th>
<th>M</th>
<th>Re</th>
<th>( \alpha )</th>
<th>( \text{Tu} (%) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td>0.40</td>
<td>(5.00 \times 10^6)</td>
<td>0.00</td>
<td>0.03</td>
</tr>
<tr>
<td>A2</td>
<td>0.40</td>
<td>(5.00 \times 10^6)</td>
<td>0.00</td>
<td>0.34</td>
</tr>
<tr>
<td>C1</td>
<td>0.40</td>
<td>(5.00 \times 10^6)</td>
<td>2.00</td>
<td>0.03</td>
</tr>
<tr>
<td>C2</td>
<td>0.40</td>
<td>(5.00 \times 10^6)</td>
<td>2.00</td>
<td>0.34</td>
</tr>
</tbody>
</table>

Figure 5.1: Flat plate grid and geometry

Figure 5.2: NACA0012 grid and geometry

Geometries are illustrated in Figures 5.1 and 5.2 for the flat plate and NACA0012 airfoil, respectively. For the remainder of this chapter, test cases will be described using the information provided by these tables. For example, test case 1A1 represents low free-stream turbulence, subsonic, transitional flow over a flat plate at 0° angle of attack, while test case 2C2 represents high free-stream turbulence, subsonic, transitional flow over a NACA0012 airfoil at a 2° angle of attack.

5.2 Solution Algorithm Parameter Selection and Modifications

Simulating flow with laminar to turbulent transition introduces substantial instabilities to the solution process, which can lead to poor convergence of the algorithm. In the early stages of convergence, the non-
linearities present in the transitional flow result in large updates to the transition and turbulence model variables, which can destabilize the solution algorithm. Specifically, during the approximate-Newton phase, large updates to the turbulence model and intermittency equation can lead to the development of large negative values for $\tilde{\nu}$ and $\gamma$. Intermittency is coupled to the production term in the Spalart-Allmaras turbulence model, presented in equation (2.62). The negative $\gamma$ values transform the production term into a negative quantity, essentially generating an additional destruction term for the Spalart-Allmaras model. The presence of an additional destruction term results in an increase in the production of negative $\tilde{\nu}$ values. These non-physical values, if left unabated, lead to negative pressure and density values in the flow. Therefore, it is critical to introduce measures that limit the solution update between nonlinear iterations to ensure that $\gamma$, and $\tilde{\nu}$, remain positive.

The following subsections present the modifications to the solution algorithm discussed in the previous chapter which were made in order to stabilize the solution and improve the robustness of the algorithm. It is important to note that the following modifications do not affect the converged steady-state flow solution, if one is obtained. The parameters introduced do not affect the residual of the governing equations, and therefore only influence convergence of a flow solution.

### 5.2.1 Time Step Considerations

In their work, Chisholm and Zingg [7] developed limits on the time step used for the turbulence model for a two-dimensional Newton-Krylov flow solution algorithm. The limits were introduced in order to maintain positive values of $\tilde{\nu}$, which would otherwise become negative after large solution updates. Their approach was to develop a local time step for the turbulence model equation. Further work on the Newton-Krylov solution algorithm by Osusky and Zingg [35] has demonstrated the need for a reduced turbulent model time step when solving flows involving laminar-turbulent transition. The work performed by Chisholm and Zingg [7] and Osusky and Zingg [35] has greatly influenced the time step approach implemented in the current work.

Three time step approaches were developed and implemented in order to stabilize the transition and turbulence model equations. The effect of these approaches on the convergence histories of the flow solution for several test cases was examined. The time step approaches and their impact on the performance of the solution algorithm are discussed in the follow subsections.

#### Reference Time Step Restriction

The reference time step used for the approximate Newton phase, given by equation (4.8), was introduced in Chapter 4. It consists of two parameters, $a$ and $b$, which determine the initial value of the time step and the growth rate, respectively. In order to ensure a robust globalization, the initial value, $a$, is given a value of 0.001. Through several test cases, Osusky and Zingg [35] determined a range of $b$ values between 1.05 and 1.5 was effective for fully turbulent flow, with $b$ limited to a value of 1.05 for explicitly tripped transitional flow. However, the instabilities introduced by the nonlinear properties of the transition model equations are slow to resolve and necessitate additional restriction of the reference time step. Simulations of the flat plate test case have revealed that the solution algorithm has difficulty determining the location of transition for the initial guess. During this phase, it is critical that the reference time step remain small. The reference time step grows exponentially with the number of outer, or nonlinear, iterations. With a $b$ value of 1.05, the time step grows to large values during the approximate Newton phase, while
The transition model is attempting to determine the location of transition. These large time step values lead to large solution updates, which destabilize the solution.

Figure 5.3 presents the global residual, turbulence, and transition model residuals, versus the nonlinear iteration index, with no time step restrictions for test case 1A1. As the nonlinear iteration index increases, the reference time step grows, leading to the generation of negative values of $\gamma$ and $\nu$, due to large solution updates. These non-physical values destabilize the solution, leading to a rapid divergence of the solution algorithm.

To address this problem, a restriction to the reference time step during the start-up phase was introduced. The restriction was developed so as to limit the value of $\Delta t_{ref}^{(n)}$ to a maximum value, $\Delta t_{max,dt}$, while the residual was above a threshold value, specified by $R_{max,dt}$. The restriction is defined as

$$\Delta t_{ref}^{(n)} = \begin{cases} \Delta t_{max,dt} : & \| R^{(n)} \|_2 \geq R_{max,dt} \\ \Delta t_{ref}^{(n)} : & \| R^{(n)} \|_2 \leq R_{max,dt} \end{cases}$$

where $\Delta t_{max,dt}$ and $R_{max,dt}$ must be optimized to perform efficiently for the majority of relevant flow conditions. Simulations were conducted using the flat plate test case in order to determine optimal values for these parameters. Figure 5.4 illustrates the effect of varying the value of the $R_{max,dt}$ parameter on the convergence behaviour of the algorithm. A $\Delta t_{max,dt}$ value of 10 was used for this initial test case simulation. This value was chosen based on the last stable time step in the previous simulation, presented in Figure 5.3. The reference time step before the solution process began to diverge represents the last stable time step. Values of $R_{max,dt}$ within an order of magnitude of 0.01 were examined, as this represents the residual where the solution began to diverge.

From Figure 5.4 it is evident that a value for $R_{max,dt}$ of 0.5 produces the quickest convergence. Larger values result in the reference time step increasing before the transition model equations are able...
to determine a suitable initial guess for the location of transition. This results in large, unstable solution updates, which cause the solution to diverge. The choice of residual values below this limit leads to poor convergence and produces excessive computational costs for the simulation. However, during the transient stages of the solution process large residual values are generated, which indicate the solution is still unstable during the start-up phase. In order to improve the robustness of the algorithm, a $R_{\text{max},dt}$ value of 0.01 was chosen for the remainder of the flat plate simulations in the current work. This value provides the solution algorithm with an optimal balance between computational cost and stability.

The effect of varying the value for $\Delta t_{\text{max},dt}$ on the convergence behaviour was examined, with the results presented in Figure 5.5. As expected, increasing the limit on the reference time step resulted in large solution updates, which destabilized the solution, while decreasing the limit delayed convergence, with a $\Delta t_{\text{max},dt}$ value of 1 significantly increasing the computational time required to obtain convergence.

In summary, $R_{\text{max},dt}$ and $\Delta t_{\text{max},dt}$ values of 0.01 and 10 were found to produce optimal convergence properties for the flat plate test case. However, additional test cases are required to find suitable values that provide stable, efficient convergence over a wider range of flow conditions.

**Turbulence and Transition Model Time Step Scaling**

For explicitly tripped flow using the Spalart-Allmaras turbulence model, Osusky and Zingg [35] examined the idea of limiting the turbulence model time step, with the mean-flow equation time steps remaining unchanged. Their approach can be represented as

$$\Delta t_{\text{SA,trip}} = \frac{1}{100} \Delta t,$$

(5.1)

where $\Delta t$ is the local time step defined by equations (4.6 and 4.7). A similar approach was investigated
in the current work for the turbulence model and transition model equations, given by

\[
\Delta t_\nu = \frac{1}{100} \Delta t,
\]
\[
\Delta t_\gamma = \frac{1}{100} \Delta t,
\]
\[
\Delta t_{\text{Ref}} = \frac{1}{100} \Delta t
\]  

(5.2)

Simulations of test case 1A1 were conducted with scaling applied to the turbulence model time step, transition model time steps, and with both the turbulence and the transition model time steps scaled. The results of these investigations are illustrated in Figure 5.6, along with the convergence history of the restricted time step with \( R_{\text{max,dt}} = 0.01 \) and \( \Delta t_{\text{max,dt}} = 10 \).

Although time step scaling was demonstrated to be effective when applied to the Spalart-Allmaras turbulence model for flow involving explicitly tripped transition, the method proves to be ineffective for flows involving transition prediction. The time step scaling reduces the local time step for the transition and turbulence model equations, which slows the solution process. However, even with the application of time step scaling the time step increases to unstable values before a suitable initial iterate can be found. Therefore, although time step scaling delays divergence, it does not adequately address the stability concerns associated with transition prediction.

Adaptive Reference Time Step

The development of non-physical variable quantities, such as negative turbulence and transition model values, can quickly destabilize the solution algorithm. Therefore, in addition to restricting the reference time step during the start-up phase of the solution, a modification was developed that scales back the
reference time step when non-physical values are detected. At each nonlinear solver update and at every node, the turbulence and transition model variables are checked to ensure positivity and that they are within their stable limits. If a non-physical value is detected, the reference time step is scaled by

\[
\Delta t^{(n+1)}_{\text{ref}} = \Delta t^{(n)}_{\text{ref}} \times 0.9,
\]

where \(\Delta t^{(n)}_{\text{ref}}\) is the reference time step at the previous nonlinear iteration. Scaling the time step when non-physical conservative variables are detected can lead to stalling of the solution algorithm if a form of variable clipping or damping is not included. This is because the reference time step is reduced at each outer iteration, which can lead to excessively small time steps if the solution algorithm is not able to quickly recover physical variable values.

The effect of the implementation of an adaptive reference time step, in combination with update damping and clipping measures, is explored in the following section.

### 5.2.2 Damping of the Solution Update

The previous simulations have demonstrated the necessity of ensuring the turbulence model and transition model variables stay within their stable limits. This includes maintaining positivity, and ensuring the variable values do not grow excessively large. As previously stated the presence of negative intermittency values can result in negative pressure and density values in the flow domain. This leads to a breakdown in the solution process. In addition, the intermittency transition model is not designed to handle negative \(\gamma\) values, and has difficulty recovering physical values once negative \(\gamma\) values are produced. In order to ensure the solution updates do not generate non-physical or excessively large values, variable clipping and solution update damping measures have been implemented. These variable limiting methods are only utilized during the start-up phase and therefore do not have an effect on the
steady-state solution.

The use of variable clipping has been widely adopted for the Spalart-Allmaras turbulence model, in order to prevent the production of negative $\tilde{\nu}$ values [35]. However, the use of variable clipping can lead to poor solution algorithm performance and stalled solutions when simulating complex flows, where the update at each nonlinear iteration continues to attempt to drive the variable outside of its limits. Therefore, in order to avoid stalled solutions and improve convergence, damping of the solution update was introduced. The solution damping is defined as

$$Q^{(n+1)} = Q^{(n)} + \delta_t \times \Delta Q^{(n)},$$

where $Q^{(n)}$ represents the value of the conserved variable at the previous iteration, $\Delta Q^{(n)}$ is the solution update, and $\delta_t$ represents the damping factor. The damping factor is specified as

$$\delta_t = 0.9^p,$$

where $p$ represents the iteration index of the damping function. The solution damping operates as follows:

1. Update solution
2. Check for turbulent model and transition model variables outside of stable range,
3. Calculate damping factor, equation (5.5),
4. Apply damping, given by equation (5.4),
5. Repeat 1-4.

Test case 2A1 was used to investigate the effect of solution update damping, versus variable clipping, on convergence behaviour. The convergence histories for this test case are presented in Figure 5.7. Stable limits for $\gamma$ and $\tilde{Re}_{\theta t}$ were developed by Langtry [21], and are defined as $\gamma \in (0, 1)$ and $\tilde{Re}_{\theta t} \geq 0$. However, it is important to note that in the case of laminar-turbulent boundary layer transition due to a laminar separation bubble the limit on $\gamma$ must be relaxed to allow $\gamma_{eff}$ to take on values larger than 1, which are produced by $\gamma_{sep}$ in equation (2.61). For these cases the limit on $\gamma$ is specified as $\gamma \in (0, 1.5)$. In addition, an upper limit of $5 \times 10^4$ was placed on $\tilde{Re}_{\theta t}$ to restrain it from growing to unstable values. In order to obtain converged solutions, the reference time restriction, discussed in Section 5.2.1, was implemented for the NACA0012 test cases, with optimal values of $R_{\text{max}, dt}$ and $\Delta t_{\text{max}, dt}$ determined to be 1 and 10, respectively.

As expected, without any variable limiting measures the solution diverges. A converged steady-state solution is obtained for both the variable clipping and solution update damping methods, with the variable clipping method requiring fewer nonlinear iterations to converge. Although variable clipping was able to achieve faster convergence, solution update damping provides increased robustness for the solution algorithm and prevents the algorithm from developing a stalled solution.

In addition to variable clipping and solution update damping, the performance of the adaptive reference time step method, introduced in the previous section, was examined. Simulations were conducted for test case 2C1 with variable clipping and solution damping with and without the adaptive reference time step modification. The convergence histories for these simulations are presented in Figure 5.8.
Figure 5.7: Convergence histories for Test Case 2A with variable limiting methods and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, respectively

Figure 5.8: Convergence histories for Test Case 2C with variable clipping and adaptive reference time step methods and $\Delta t_{\text{max,dt}}$ and $R_{\text{max,dt}}$ values of 10 and 1, respectively

The convergence behaviour of the variable clipping and solution update damping methods coupled with the adaptive reference time step demonstrate the effectiveness of these modifications. For both variable limiting methods the computational time required to obtain a converged steady-state solution was reduced when the adaptive reference time step method was applied. Furthermore, the convergence history...
Figure 5.9: Convergence histories for Test Case 1A2 residual reduction study and $\Delta t_{max,dt}$ and $R_{max,dt}$ values of 10 and 0.01, respectively.

indicates a smoother, more stable convergence profile. The results of these simulations demonstrate the increase in effectiveness of the solution update damping variable limiting method over variable clipping as the complexity of the flow is increased. However, for simulations involving less complex flow the application of solution update damping and an adaptive reference time step increased the computational cost. For the current work variable clipping was utilized as the primary method for limiting the turbulence and transition model variables. Solution update damping was applied for cases involving complex flows and when variable clipping demonstrated difficulties achieving convergence. An adaptive reference time step was used for both cases to improve robustness.

5.2.3 Switch from Approximate to Inexact-Newton Phase

The transition from the approximate-Newton phase to the inexact-Newton phase is an important parameter. As discussed in Section 4.1.3, the switch between the phases is initiated when the value of the nonlinear residual drops below a threshold value. Switching to the inexact-Newton phase before an accurate initial guess is found could lead to divergence of the solution algorithm, or poor performance of the inexact-Newton phase, while delaying the switch too long could lead to the stalling of the approximate-Newton phase.

Cases 1A2 and 2A2 are used to investigate the effect of the threshold value for the approximate to inexact-Newton switch on the performance of the flow solver. The residual reduction threshold is varied from a value of $1 \times 10^{-2}$ to a value of $1 \times 10^{-6}$. Osusky and Zingg [35] conducted a similar study to investigate the performance of different threshold values for fully turbulent and tripped turbulence flows. They concluded a value for $R_d$ of $1 \times 10^{-4}$ was optimal. The focus of this study is to determine if the algorithm with the addition of the transition model equations requires a different threshold value. The convergence history for case 1A2 is illustrated in Figure 5.9, while Figure 5.10 presents the convergence
Figure 5.10: Convergence histories for Test Case 2A residual reduction study and $\Delta t_{\text{max},dt}$ and $R_{\text{max},dt}$ values of 10 and 1, respectively

history for case 2A.

The results of test case 1A indicate the solution process failed to converge for residual reduction thresholds of $1 \times 10^{-2}$ and $1 \times 10^{-3}$. This is a consequence of the solution algorithm switching to the inexact-Newton phase before a suitable initial iterate was found. The solution algorithm is able to obtain converged steady-state solutions when the threshold value is decreased. However, the computational time required for convergence of the solution process is increased for threshold values lower than $R_d = 10^{-4}$.

For test case 2A, a converged solution was obtained for residual reduction values of $1 \times 10^{-5}$ and $1 \times 10^{-6}$. The solution failed at a residual reduction of $1 \times 10^{-2}$, and stalled at threshold values of $1 \times 10^{-3}$ and $1 \times 10^{-4}$, as the approximate Newton phase failed to produce a suitable initial iterate for the inexact-Newton method. Based on these findings, a residual reduction value of $1 \times 10^{-5}$ was used for the simulations conducted in the remainder of this work, as it provides a good balance between speed and robustness.

5.3 Negative Turbulence Model Formulation

Allmaras et al. [3] proposed a modification to the original Spalart-Allmaras turbulence model which allows for negative values of $\tilde{\nu}$. Although these values are non-physical, they often appear in transient states during the solution process, or as a result of large solution updates. This modification is an alternative to the default method of clipping the Spalart-Allmaras model variable to trim any negative values of $\tilde{\nu}$. The modified terms in the 'negative' Spalart-Allmaras turbulence model are given by

\[
\tilde{S} = \begin{cases} 
    \text{Re}(S + \tilde{S}) & : \tilde{S} \geq -c_{v2}S \\
    \text{Re}(S + \frac{S(c_{v2}^2 S + c_{v3} \tilde{S})}{(c_{v3} - 2c_{v2})S - \tilde{S}}) & : \tilde{S} < -c_{v2}S
\end{cases}
\]
where $c_{v2}$ and $c_{v3}$ are equal to 0.7 and 0.9, respectively, while the parameter $\bar{S}$ is defined as

$$
\bar{S} = \frac{\tilde{\nu}}{\kappa^2 d^2 f_{v2}}.
$$

Additionally, the production, destruction, and diffusion terms of the Spalart-Allmaras model are modified according to

$$
P_{\tilde{\nu}} = \begin{cases} 
\frac{c_{w1}}{Re} [1 - f_{t2}] \tilde{S} \tilde{\nu} & : \tilde{\nu} \geq 0 \\
\frac{c_{w1}}{Re} [1 - c_{t3}] \tilde{S} \tilde{\nu} & : \tilde{\nu} < 0,
\end{cases}
$$

$$
D_{\tilde{\nu}} = \begin{cases} 
\frac{1}{Re} c_{w1} f_w - \frac{c_{w1} f_{t2}}{\kappa} \left( \frac{\tilde{\nu}^2}{\tilde{\nu}} \right) & : \tilde{\nu} \geq 0 \\
\frac{1}{Re} c_{w1} \left( \frac{\tilde{\nu}^2}{\tilde{\nu}} \right) & : \tilde{\nu} < 0,
\end{cases}
$$

and

$$
\text{diffusion} = \begin{cases} 
\frac{1+c_{n1}}{\sigma Re} \nabla \cdot [(\nu + \tilde{\nu}) \nabla \tilde{\nu}] - \frac{c_{n1}}{\kappa Re} (\nu + \tilde{\nu}) \nabla^2 \tilde{\nu} & : \tilde{\nu} \geq 0 \\
\frac{1}{\sigma Re} \nabla \cdot [(\nu + f_n \tilde{\nu}) \nabla \tilde{\nu}] - \frac{c_{n2}}{\kappa Re} \{ \nabla \cdot (\nu + \tilde{\nu}) \nabla \tilde{\nu} - (\nu + \tilde{\nu}) \nabla^2 \tilde{\nu} \} & : \tilde{\nu} < 0,
\end{cases}
$$

where the constant $c_{n1}$ is given a value of 16. The eddy viscosity variable $\mu_t$ is modified to be equal to zero when a negative value of $\tilde{\nu}$ is produced. The modifications to the production, destruction, diffusion, and vorticity terms are only active in regions of negative $\tilde{\nu}$, which for our current work is only true during transient stages in the solution process. Therefore, this modification should not affect the converged steady-state solution.

The performance of the negative Spalart-Allmaras model formulation was investigated in a simulation of test case 2C2, with the convergence histories of the algorithm with and without the modification illustrated in Figure 5.11. For these simulations $\Delta t_{\text{max},dt}$ and $R_{\text{max},dt}$ values of 1 and 10 were used, with a threshold value of $1 \times 10^{-5}$, and variable clipping with an adaptive reference time step.

Without the negative Spalart-Allmaras model modification, the solution stalls at a residual of approximately $10^{-1}$. The results of this simulation indicate the Spalart-Allmaras variable, $\tilde{\nu}$, is being clipped to prevent it from going negative at several nodes within the flow domain, at every nonlinear iteration, which leads to the stalled solution. The solution algorithm with the negative Spalart-Allmaras model formulation is able to obtain a converged steady-state solution after approximately 1100 iterations. The results of this test case demonstrate the effectiveness of the Spalart-Allmaras model modification in handling transient solutions involving negative $\tilde{\nu}$ values. These transient states appear often for simulations involving transition prediction, and therefore this modification has been applied in the majority of the simulations presented in the current work.

### 5.4 Computational Cost

The goal of this chapter was to provide parameters and introduce modifications to the algorithm that result in an efficient, robust, and accurate flow solution algorithm. To investigate the efficiency of the chosen parameters and modifications, a simulation of test case 2A1 was conducted both with and without transition prediction. For this study $\Delta t_{\text{max},dt}$ and $R_{\text{max},dt}$ values of 1 and 10 were used with a threshold
Figure 5.11: Convergence histories for Test Case 2C2 with negative Spalart-Allmaras model modification, Δt_{max,dt} and R_{max,dt} values of 10 and 1, variable clipping, an adaptive reference time step, and an R_d value of $1 \times 10^{-5}$ and variable clipping with an adaptive reference time step. For both cases the initial time step value, $a$, is given a value of 0.001, with $b$ limited to a value of 1.05 for both cases. Figure 5.12 illustrates the convergence history for both cases. The total number of nonlinear outer and linear inner iterations for each case is presented in Table 5.3. The processors used in this study consist of four Intel Xeon (Sandybridge) E5-2650 CPUs, each with eight cores clocked at 2.0 GHz, resulting in a total of 32 processor cores. These results demonstrate the flow solver with transition prediction capabilities is approximately four times more computationally expensive than a simulation with a fully turbulent flow approximation, requiring significantly more linear iterations.
Figure 5.12: Convergence histories for Test Case 2A with and without transition prediction, $\Delta t_{\text{max},dt}$ and $R_{\text{max},dt}$ values of 10 and 1, respectively, variable clipping, an adaptive reference time step, and an $R_d$ value of $1 \times 10^{-5}$.

Table 5.3: Convergence performance parameters for Test Case 2A with and without transition prediction, $\Delta t_{\text{max},dt}$ and $R_{\text{max},dt}$ values of 10 and 1, respectively, variable clipping, an adaptive reference time step, and an $R_d$ value of $1 \times 10^{-5}$. Reference time step values of $a = 0.001$ and $b = 1.05$ were used for both cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>Newton Phase</th>
<th>Nonlinear (Outer) iterations</th>
<th>Linear (Inner) iterations</th>
<th>Time(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fully Turbulent</td>
<td>Approximate</td>
<td>323</td>
<td>4095</td>
<td>102.6</td>
</tr>
<tr>
<td></td>
<td>Inexact</td>
<td>6</td>
<td>1254</td>
<td>22.6</td>
</tr>
<tr>
<td>Transition Prediction</td>
<td>Approximate</td>
<td>498</td>
<td>9600</td>
<td>355.0</td>
</tr>
<tr>
<td></td>
<td>Inexact</td>
<td>37</td>
<td>6541</td>
<td>138.5</td>
</tr>
</tbody>
</table>
Chapter 6

Results

The previous chapter introduced modifications and efficient parameters for the solution algorithm which are necessary in order to obtain an efficient and robust solution algorithm. The focus of this chapter is to present results that demonstrate the accuracy of the resulting algorithm with respect to predicting the location of laminar-turbulent boundary layer transition. The accuracy of the algorithm will be examined using a series of two-dimensional verification and validation cases, with the results produced by the flow solver compared to experimental values.

The results in this section were obtained using $\Delta t_{max,dt}$ and $R_{max,dt}$ values of 1 and 10, with a threshold value of $1 \times 10^{-5}$. Variable clipping was used with an adaptive reference time step. The initial time step value, $a$, was given a value of 0.001, with $b$ limited to a value of 1.05, while a first-order upwind spatial discretization was used for the convective terms in the turbulence and transition model equations. The negative Spalart-Allmaras turbulence model was used as well. These parameters and modifications were selected based on their robustness and efficiency, as demonstrated in Chapter 5.

6.1 Flat Plate Simulations

Flat plate simulations were conducted in order to verify the accuracy of the current algorithm with the $\gamma$ - $Re_{\theta t}$ - SA transition model. The flat plate grid and geometry used in the previous chapter will be applied in the current section as well, with the details of the grid and geometry presented in Figure 5.1 and Table 5.1. The results of these simulations were compared with experimental values from a series of zero-pressure-gradient flat-plate test cases. The zero-pressure-gradient flat-plate experimental results, obtained by Schubauer and Klebanoff [45], and Schubauer and Skramstad [46] over a range of turbulence intensities will be used in the current work. The results from these experiments, specifically the effect of turbulence intensity on the approximate experimental transition onset locations, are presented in Table 6.1. As the turbulence intensity increases, the Reynolds number, $Re_x$, of transition decreases, signifying an upstream shift of the transition onset location. This is the result of the increased free-stream turbulence convecting and diffusing into the boundary layer, introducing instabilities in the laminar velocity profile, which eventually lead to turbulent spots and fully turbulent flow.

Simulations were conducted using the flat plate test case with the turbulence intensity values listed in Table 6.1. The calculated skin friction coefficient profiles for these cases are illustrated in Figure 6.1. The results demonstrate the transition model is able to predict natural transition at low turbulence intensity,
Table 6.1: Experimental location of transition onset for zero-pressure-gradient flat plate [46]

<table>
<thead>
<tr>
<th>Tu (%)</th>
<th>Re_x \times 10^{-6}</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.34</td>
<td>1.4</td>
</tr>
<tr>
<td>0.26</td>
<td>1.8</td>
</tr>
<tr>
<td>0.20</td>
<td>2.1</td>
</tr>
<tr>
<td>0.10</td>
<td>2.7</td>
</tr>
<tr>
<td>0.042</td>
<td>2.8</td>
</tr>
<tr>
<td>0.03</td>
<td>2.8</td>
</tr>
</tbody>
</table>

Figure 6.1: Skin-friction profiles for zero-pressure-gradient flat-plate, Medida and Baeder correlation

Tu = 0.03% and Tu = 0.042%, with great accuracy. The location of transition onset is determined by the location at which the skin friction coefficient begins to increase, deviating from a purely laminar profile. However, as the level of turbulence increases, beyond a turbulence intensity value of 0.042%, the model becomes less accurate, predicting the onset of transition upstream of the experimental value at a Tu value of 0.10%. As the turbulence level further increases, the algorithm regains its accuracy, predicting the locations of transition for Tu ∈ (0.20, 0.34) similar to the experimental values. The purpose of this thesis is to examine the performance of the γ - \( \tilde{Re}_\theta \) transition model when applied to flows around aerodynamic bodies, such as wings and airfoils, where free-stream turbulence intensities are low. The remainder of the verification and validation test cases conducted in this chapter operate in low free-stream turbulence flow, where Tu = 0.03% [50, 51]; therefore this inconsistency should not affect the accuracy of the remaining simulations.

As mentioned in Section 2.3.1, the critical momentum thickness Reynolds number, \( Re_{\theta_c} \), empirical
correlation developed by Medida and Baeder [30] and given by equation (2.46) was used for the current work. The accuracy of the $Re_{\theta_c}$ correlation developed by Langtry [21] for use with the SST $k - \omega$ turbulence model and given by equation (2.45) was also investigated. Although this method was developed for use with the two-equation turbulence model, previous work has been conducted [47] using this correlation implemented with the transition model coupled to the one-equation Spalart-Allmaras turbulence model. The zero-pressure-gradient flat-plate results obtained using Langtry’s correlations are illustrated in Figure 6.2. The transition onset locations calculated using the $Re_{\theta_c}$ empirical correlation developed by Langtry [21] predict transition downstream of the experimental values over the range of turbulence intensities. The critical momentum thickness Reynolds number determines the location at which the intermittency first begins to grow. This location is upstream of the actual transition onset location, as experiments have shown intermittency develops upstream of transition onset. Therefore, verifying the accuracy of the $Re_{\theta_c}$ empirical correlation is critical to ensuring the transition model calculates the correct location of transition onset. The results of this test case demonstrate the need to employ empirical correlations developed specifically for the transition model when coupled to the SA turbulence model.

Based on these findings, the choice of empirical correlations employed in the transition model, including the $Re_{\theta_c}$ empirical correlation developed by Medida and Baeder [30], which is implemented in the current work, is validated for the flow characteristics investigated in this thesis.
Chapter 6. Results

6.2 Two-Dimensional Airfoils

In addition to the zero-pressure-gradient flat-plate test cases, simulations of two-dimensional, natural-laminar-flow airfoils were conducted, with the results presented in the following section. Specifically, the NLF0416 and S809 natural-laminar-flow airfoils developed by Somers [50, 51] were investigated. The primary methods of transition for these airfoils are natural transition and separation-induced transition. The grids and geometries for these airfoils are presented in Table 6.2 and illustrated in Figures 6.3 and 6.4, respectively.

6.2.1 NLF-0416 Airfoil

The NLF0416 airfoil is a natural-laminar-airfoil, meaning it can achieve significant extents of laminar flow (≥ 30% chord) solely through favourable pressure gradients, with no boundary-layer suction or cooling. The airfoil was designed to produce high maximum lift coefficients at low speed, with reduced cruise drag coefficients. Experimental results of the NLF0416 airfoil were obtained in the low turbulence pressure tunnel at NASA Langley Research Centre [50]. Transition locations were measured for several angles of attack at a Mach number of 0.1, and a Reynolds number of $4 \times 10^6$. The turbulence intensity in the wind tunnel was measured at around $Tu = 0.03\%$. Simulations were conducted using the $\gamma - Re_{\theta t}$

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Topology</th>
<th>Blocks</th>
<th>Nodes</th>
<th>Off-wall spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>NLF0416</td>
<td>H</td>
<td>48</td>
<td>220,400</td>
<td>$5.66 \times 10^{-6}$</td>
</tr>
<tr>
<td>S809</td>
<td>H</td>
<td>48</td>
<td>220,400</td>
<td>$6.73 \times 10^{-6}$</td>
</tr>
</tbody>
</table>

Figure 6.3: Grid and geometry for NLF0416 airfoil [50]


Table 6.3: Experimental and computational locations of transition onset for NLF0416 airfoil at positive angles of attack [50]

<table>
<thead>
<tr>
<th>Angle of Attack (°)</th>
<th>(x/c)\text{upper,exp}</th>
<th>(x/c)\text{lower,exp}</th>
<th>(x/c)\text{upper,comp}</th>
<th>(x/c)\text{lower,comp}</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td>0.35 - 0.40</td>
<td>0.50 - 0.55</td>
<td>0.41</td>
<td>0.59</td>
</tr>
<tr>
<td>2.03</td>
<td>0.30 - 0.35</td>
<td>0.60 - 0.65</td>
<td>0.33</td>
<td>0.64</td>
</tr>
<tr>
<td>4.07</td>
<td>0.25 - 0.30</td>
<td>0.60 - 0.65</td>
<td>0.30</td>
<td>0.66</td>
</tr>
<tr>
<td>6.09</td>
<td>0.20 - 0.25</td>
<td>0.65 - 0.70</td>
<td>0.19</td>
<td>0.68</td>
</tr>
</tbody>
</table>

The calculated skin-friction coefficient profiles, determined for positive angles of attack, are illustrated in Figure 6.5.

The approximate experimental values for the location of transition onset and the computed values are presented in Table 6.3. Note that experimental values were only determined for positive angles of attack. As the angle of attack is increased, the upper surface location of transition onset moves forward, while the lower surface transition onset location moves further downstream. The results obtained from the application of the $\gamma - \tilde{Re}_{\theta_t}$ - SA transition model indicate a similar trend. The transition locations at 0.01° angle of attack are slightly downstream of the experimental locations. As the angle of attack is increased to 1.01° the transition location on the upper surface of the NLF0416 airfoil appears to decrease slightly; however, the results obtained at angles of attack of 2.03° and greater are similar to the experimentally determined values.

In the design of the NLF0416 airfoil, Somers noted laminar-turbulent transition occurred on the lower surface of the airfoil for all positive angles of attack and at a Reynolds number of $4.0 \times 10^6$ due to the presence of a laminar separation bubble [50]. The results produced by the $\gamma - \tilde{Re}_{\theta_t}$ transition model coupled with the SA turbulence model accurately predict the location of transition for these cases.
The point of flow separation is defined as the location where the wall shear stress on the surface of the airfoil becomes zero, and eventually negative. The results of the simulation, presented in Figure 6.5 demonstrate that the skin friction coefficient $c_f$ approaches a value of zero on the lower surface of the airfoil, for all positive angles of attack. Following transition, the turbulence in the flow increases as the flow develops into a turbulent boundary layer. This is represented by the steep rise in the skin friction coefficient downstream of the transition location.

The skin friction coefficient profiles for the NLF0416 airfoil, calculated for negative angles of attack, are presented in Figure 6.6. Although experimental values for the location of transition onset at negative angles of attack are not available for the NLF0416 airfoil, computational results were obtained using the $\gamma$ - $Re_{\theta_t}$ - SA transition model in the work performed by Schucker et al. [47]. These transition onset location values are presented in Table 6.4. The transition onset locations calculated in the current work are similar to the values calculated by Schucker [47]. Angles of attack outside of this range were examined; however converged steady-state solutions could not be obtained. As the magnitude of the angle of attack is increased, for both positive and negative angles of attack, a separation bubble begins to grow and become unstable. The location of the separation bubble oscillates. According to Schucker [47], this periodic behaviour is due to the interaction of the separation bubble with the separation-induced transition location. This feature is an unsteady effect inherent in the system of transport equations for the transition model.
Figure 6.6: Skin-friction profiles for NLF0416 airfoil at negative angles of attack

Table 6.4: Location of transition onset for NLF0416 airfoil at negative angles of attack calculated by Schucker [47] and in the current work

<table>
<thead>
<tr>
<th>Angle of Attack (°)</th>
<th>(x/c)$_{\text{upper,schuck}}$</th>
<th>(x/c)$_{\text{lower,schuck}}$</th>
<th>(x/c)$_{\text{upper,comp}}$</th>
<th>(x/c)$_{\text{lower,comp}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1.02</td>
<td>0.45</td>
<td>0.53</td>
<td>0.46</td>
<td>0.54</td>
</tr>
<tr>
<td>-2.04</td>
<td>0.46</td>
<td>0.39</td>
<td>0.47</td>
<td>0.34</td>
</tr>
<tr>
<td>-3.06</td>
<td>0.48</td>
<td>0.32</td>
<td>0.49</td>
<td>0.30</td>
</tr>
</tbody>
</table>

6.2.2 S809 Airfoil

Similar to the NLF0416, the S809 airfoil is a natural-laminar-flow airfoil, designed by Somers [51]. The airfoil was designed and tested in the low-turbulence wind tunnel of the Delft University of Technology Low Speed Laboratory. The primary objective of the S809 airfoil was to obtain a maximum lift coefficient that does not decrease as transition approaches the leading edge of the upper and lower surfaces and to achieve low profile-drag coefficients for low Reynolds number flow, over a range of angles of attack. Experimental results for the S809 airfoil were obtained at a Mach number of 0.1 and Reynolds number of $3.0 \times 10^6$. The free-stream turbulence intensity in the wind tunnel varied between $T_u = 0.02\%$ and $T_u = 0.04\%$.

For a Reynolds number of $3.0 \times 10^6$ and an angle of attack of $0^\circ$, Somers noted the transition mechanism on the upper surface of the airfoil is a laminar separation bubble [51]. As the angle of attack increases, the separation bubble decreases in length, disappearing completely at an angle of attack of $4.0^\circ$. Moreover, as the angle of attack increases, from a value of $-1.0^\circ$, the upper surface location of
transition onset moves forward, while the transition onset location on the lower surface of the airfoil moves downstream. Skin-friction profiles for the S809 airfoil were calculated using the $\gamma - \tilde{Re}_\theta$ transition model. Table 6.5 presents the calculated and experimental transition onset locations, while the results are illustrated in Figure 6.7. A free-stream turbulence intensity of $Tu = 0.03\%$ was used for these simulations. This value represents an average of the free-stream turbulence intensity levels present in the wind tunnel which was used to obtain the experimental results. The calculated skin-friction coefficient profiles for the S809 airfoil indicate the predicted transition onset locations match the experimental values for all angles of attack. Furthermore, the experimental separation-induced transition onset locations evident on both the upper and lower surfaces of the airfoil are similar to the computational values.

Simulations conducted at larger angles of attack failed to converge due to stability issues. It was noted by Somers [51] that as the angle of attack was increased, turbulent flow separation began to develop on
the upper surface of the airfoil at the trailing edge. This separation introduces unsteady effects to the flow solution, resulting in the algorithm being unable to produce a steady-state flow solution.

6.3 Two-Dimensional Multi-Element High-Lift Configurations

Multi-element airfoil configurations consisting of leading edge slats and/or trailing-edge flaps are frequently used in order to increase maximum lift and delay the onset of stall at low speeds. These configurations produce complex flows, with turbulence generated by the wake of leading elements strongly affecting the location of laminar-turbulent transition of the downstream elements. It is important that the current implementation of the $\gamma - \tilde{Re}_t$ transition model be able to simulate these effects.

6.3.1 NLR-7301 Two-Element Airfoil

The NLR-7301 airfoil, designed at the Netherlands Aerospace Laboratory (NLR) [1], was selected based on the significant amount of experimental and computational data available. The two-element airfoil is a thick supercritical airfoil which consists of a blunt main element with a trailing-edge flap. The grid and geometry for this airfoil is illustrated in Figure 6.8 and presented in Table 6.6.

To replicate experimental conditions, the NLR-7301 airfoil was simulated over a range of Mach numbers with a Reynolds number of $2.2 \times 10^6$, and at an angle of attack of $0.85^\circ$. The experimental turbulence intensity of the NLR Pilot Tunnel in which the experiments were conducted was not provided. Wang and Mian [60] conducted a computational study of the NLR-7301 airfoil using the original $\gamma - \tilde{Re}_t$ transition model developed by Langtry [21], coupled to the $k - \omega$ SST turbulence model, in an
Table 6.6: Grid properties for NLR-7301 airfoil

<table>
<thead>
<tr>
<th>Topology</th>
<th>Blocks</th>
<th>Nodes</th>
<th>Off-wall spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>H</td>
<td>8</td>
<td>33,075</td>
<td>$2.06 \times 10^{-5}$</td>
</tr>
</tbody>
</table>

unstructured finite-volume Reynolds-averaged Navier-Stokes based solver (HUNS3D). In this study they determined that a value of $T_{u\infty} = 0.30\%$ produced results similar to the experimental values. Therefore, a turbulence intensity value of $0.30\%$ was used to obtain the following results.

Skin-friction profiles for the main element of the two-element airfoil, calculated using the $\gamma - \tilde{R}e_{\theta t}$ transition model, are illustrated in Figure 6.9, while Figure 6.10 illustrates the calculated skin-friction profile for the trailing-edge flap. The calculated transition onset locations for the main element, compared with the locations determined through experiment [1] and calculated using the $\gamma - \tilde{R}e_{\theta t}$ transition model coupled with the $k - \omega$ SST turbulence model [60], are listed in Table 6.7. No experimental results for the transition locations of the trailing-edge flap were provided, and they are not discussed in the work of Wang and Mian [60].

The results demonstrate the $\gamma - \tilde{R}e_{\theta t}$ coupled with the Spalart-Allmaras turbulence model is able to accurately determine the transition location on the lower surface of the main element of the airfoil; however, it encounters difficulty when determining the transition location of the upper surface of the airfoil. This error may be due to the coarse mesh spacing around the blunt leading edge of the airfoil. In the work of Schucker [47] it was noted that the laminar-turbulent transition location moved upstream with increasing mesh coarseness. It should be noted that the mesh used in the work of Wang and Mian [60] consisted of over 190,000 nodes, compared to the 33,075 nodes used in this study. Therefore, additional simulations using finer grids are required to determine if this error is associated with the transition model or due to the mesh spacing.
Figure 6.9: Calculated skin-friction profiles for the main element NLR-7301 airfoil over a range of Mach numbers

Figure 6.10: Calculated skin-friction profiles for the trailing-edge flap of the NLR-7301 airfoil over a range of Mach numbers
Table 6.7: Computed locations of transition onset on main element of NLR-7301 airfoil over a range of Mach numbers with $Re = 2.2 \times 10^6$, $Tu_\infty = 0.30\%$, and an angle of attack of $0.85^\circ$.

<table>
<thead>
<tr>
<th>Mach Number</th>
<th>$(x/c)<em>{u</em>{\text{exp}}}$</th>
<th>$(x/c)<em>{l</em>{\text{exp}}}$</th>
<th>$(x/c)<em>{u</em>{\text{wang}}}$</th>
<th>$(x/c)<em>{l</em>{\text{wang}}}$</th>
<th>$(x/c)<em>{u</em>{\text{comp}}}$</th>
<th>$(x/c)<em>{l</em>{\text{comp}}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.20</td>
<td>0.18</td>
<td>0.60</td>
<td>0.17</td>
<td>0.60</td>
<td>0.05</td>
<td>0.64</td>
</tr>
<tr>
<td>0.30</td>
<td>0.15</td>
<td>0.59</td>
<td>0.16</td>
<td>0.59</td>
<td>0.06</td>
<td>0.59</td>
</tr>
<tr>
<td>0.40</td>
<td>0.14</td>
<td>0.58</td>
<td>0.14</td>
<td>0.58</td>
<td>0.06</td>
<td>0.58</td>
</tr>
</tbody>
</table>
Chapter 7

Conclusions and Recommendations

The following section provides a summary of the thesis objectives and conclusions with recommendations for future work provided in Section 7.2.

7.1 Conclusions

Through the course of this work, a parallel Newton-Krylov-Schur algorithm with transition prediction capabilities provided by the $\gamma - \tilde{R}c_{\theta t}$ transition model [21] coupled to the Spalart-Allmaras turbulence model was developed. Modifications to the transition model developed by Schucker [47] and Medida and Baeder [30] were examined, with refinements from their work implemented in order to maximize the accuracy and efficiency of the current flow solver. Spatial discretization of the Navier-Stokes, Spalart-Allmaras turbulence model, and transition model equations was achieved using an SBP-SAT discretization strategy. To increase the efficiency of simulations the computational domain was decomposed into multiple blocks, resulting in multi-block structured grids, which allow for parallel computation of the domain, and reduces the computational time required for each simulation. A Newton-Krylov solution algorithm making use of a pseudo-transient continuation strategy was applied to the nonlinear set of equations produced by the discretization of the governing equations to drive the solution from an initial guess to a converged steady-state solution. The large system of linear equations generated at each Newton iteration was solved using the GMRES linear solver, preconditioned using an approximate-Schur parallel preconditioner.

The stability and robustness of the solution algorithm was optimized in order to obtain efficient and reliable convergence. This optimization process included the implementation of modifications to the globalization strategy, the selection of efficient solution algorithm parameters, and investigated the performance of the negative Spalart-Allmaras turbulence model formulation. Large updates to the turbulence model and intermittency equations were found to produce non-physical values for these variables, which destabilized the solution process. A restriction to the reference time step was introduced to limit these solution updates. In addition, variable clipping and solution update damping methods were found to be effective for ensuring the variables remained within their stable bounds. For complex flows the use of an adaptive reference time step was shown to produce a more stable convergence history. Optimal threshold values for the switch between the approximate and inexact-Newton phases were found. A first-order upwind strategy was applied to the advection terms for the turbulence and transition model
equations, which improved the stability of the algorithm significantly. In addition, the performance of a negative Spalart-Allmaras model formulation was investigated and demonstrated to be effective for solving the transient states produced during simulation of flows involving transition.

The purpose of this work was to investigate the performance of a two-equation $\gamma - Re_{\theta_t}$ transition model based on differential transport equations formulated with empirical correlations coupled to the Spalart-Allmaras turbulence model in a three-dimensional multi-block structured finite-difference solver. The goal was to produce an algorithm which can accurately and efficiently predict transition due to natural transition and separation-induced transition. The performance of the Newton-Krylov-Schur algorithm was investigated through the application of several two-dimensional verification and validation cases. Simulations of flat plate test cases demonstrate the $\gamma - Re_{\theta_t}$ - SA transition model is able to predict locations of natural transition which match experimental results. Furthermore, the accuracy of the transition model in determining transition due to a laminar separation bubble was examined by calculating the flow field over two-dimensional, natural-laminar-flow airfoils. The physical phenomena demonstrated in experimental wind tunnel studies of these airfoils were reproduced in the computational investigations. In addition the experimental transition onset locations due to separation-induced transition were predicted accurately using the transition model. The results demonstrate the $\gamma - Re_{\theta_t}$ - SA transition model is able to accurately predict the location of transition onset due to both natural and separation-induced transition, and model the complex flow regimes present during boundary-layer transition. Furthermore, simulations of a multi-element, high-lift airfoil configuration explored the accuracy of the algorithm in predicting the locations of transition for flows involving multiple lifting surfaces, and demonstrate the robustness of the solution algorithm.

In conclusion, the results demonstrate the Newton-Krylov-Schur algorithm with transition prediction capabilities is able to accurately predict laminar-turbulent boundary-layer transition based on two-dimensional boundary-layer theory, for the simulations considered in this work. Furthermore, the modifications made to the solution algorithm formulation provide efficient and robust convergence of the flow solution for a wide range of flow conditions. Therefore, based on these findings, the $\gamma - Re_{\theta_t}$ transition model, coupled to the Spalart-Allmaras turbulence model in a three-dimensional multi-block structured finite-difference solver, has been demonstrated to be an effective method for predicting laminar-turbulent boundary-layer transition due to two-dimensional effects.

7.2 Recommendations

Several recommendations can be made based on the findings of the current work in order to improve the applicability and efficiency of the existing algorithm.

Practical engineering studies often involve the study of large, complex, three-dimensional aerodynamic bodies. In order to include the effects of boundary-layer transition in these studies it is necessary to extend the current transition model to incorporate the effects of three-dimensional boundary-layer instabilities on the location of laminar-turbulent transition. Specifically, the implementation of correlations for predicting transition due to crossflow instabilities, such as those developed by Medida and Baeder [31], would significantly increase the applicability of the current solution algorithm.

Furthermore, in order to ensure that the flow solver is able to converge efficiently to a steady-state solution it is recommended that the algorithm optimization strategy, discussed in Chapter 5, be expanded to include a wider range of flow conditions. Transonic flow regimes are of particular interest, as they
represent the flow characteristics of commercial aircraft at cruise. Specifically, the optimization of the algorithm for three-dimensional flows is recommended. In addition, the development of a global set of input parameters for the modifications discussed in Chapter 5 which can accurately and efficiently model two and three-dimensional subsonic and transonic flows would significantly decrease user input, and increase the usability of the algorithm.

Chapter 6 revealed the solution algorithm encountered difficulties with respect to solving two-dimensional flows with large levels of flow separation. Even though the solution algorithm was able to accurately simulate flow separation at low angles of attack, to improve the robustness of the solver it is suggested that further investigation into the convergence histories of these cases be conducted, with methods for overcoming these instabilities associated with the transition model be developed.

Finally, the integration of the $\gamma$ - $Re_{\theta t}$ transition model, coupled to the Spalart-Allmaras turbulence model, with an aerodynamic shape optimization algorithm could lead to the development of novel aircraft designs, which make use of the favourable properties of laminar boundary layers.
References


