A Validation Study of SC/Tetra CFD Code

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Engineering

by

Hongtao Yu
B.E., Dalian Jiaotong University, 2011

2014
Wright State University
GRADUATE SCHOOL

April 2, 2014


______________________________
George P.G. Huang, Ph.D.
Thesis Director

______________________________
George P.G. Huang, Ph.D., Chair
Department of Mechanical and Materials Engineering
College of Engineering and Computer Science

Committee on Final Examination

______________________________
George P.G. Huang, Ph.D.

______________________________
Zifeng Yang, Ph.D.

______________________________
Joseph Shang, Ph.D.

______________________________
Robert E. W. Fyffe, Ph.D.
Vice President of Research and
Dean of the Graduate School
ABSTRACT


The goal of this study is to validate the accuracy of the SC/Tetra CFD code by modeling four cases: a 2D zero pressure gradient flat plate, an axisymmetric separated boundary layer, flow over an NACA0012 airfoil, and flow over an NACA 4412 airfoil trailing edge separation. The results will be compared to experimental data and CFL3D results. This study also investigates the sensitivity of SC/Tetra results to different setup details such as mesh resolution, the turbulence model, and under-relaxation settings. Automation, using the Visual Basic programming language, was used to vary these setup details and efficiently process a large number of simulations for the NACA 0012 case. The Flat plate case focuses on skin friction coefficient and velocity profiles. The axisymmetric separated boundary layer case concentrates on pressure coefficient and velocity profiles. For the NACA 0012 airfoil, evaluation focuses on Cp profiles. Cp and velocity profiles and streamlines are considered for the NACA 4412 airfoil. In general, coarser meshes are shown to be less accurate than finer meshes and the SST k-ω turbulence model is more accurate than others.
Table of Contents

1 Introduction ...........................................................................................................1
  1.1 Motivation ........................................................................................................1
  1.2 Four Analyzed Cases .....................................................................................3
    1.2.1 2D Zero Pressure Gradient Flat Plate Validation .........................................3
    1.2.2 Axisymmetric Separated Boundary Layer ....................................................3
    1.2.3 Flow over NACA0012 Airfoil ....................................................................4
    1.2.4 Flow over NACA4412 Airfoil ....................................................................5

2 Methodology .........................................................................................................6
  2.1 Introduction to the SC/Tetra Software .............................................................6
  2.2 The Law of the Wall .......................................................................................7
  2.3 Turbulence Models .........................................................................................9
    2.3.1 The Shear-Stress Transport (SST) k-ω Turbulence Model .............................9
    2.3.2 The Spalart-Allmaras (SA) Turbulence Model .............................................11
    2.3.3 The Standard k-ε Turbulence Model .........................................................13
    2.3.4 The Abe-Nagano-Kondoh (AKN) Turbulence Model ...............................13

3 2D Zero Pressure Gradient Flat Plate ...............................................................15
  3.1 Problem Overview ..........................................................................................15
  3.2 Simulation Set up ...........................................................................................15
    3.2.1 Structured Meshes ......................................................................................15
    3.2.2 Analysis Conditions ....................................................................................17
  3.3 SST Turbulence Model Results and Conclusions .........................................18
    3.3.1 Additional Results: Changing Inflow Turbulence Properties to Fix the Transition Problem for the SST Turbulence Model ..................................................27
  3.4 SA Turbulence Model Results and Conclusions .........................................28
    3.4.1 Additional Results: Changing Inflow Turbulence Properties to Fix the Transition Problem for the SA Turbulence Model ..................................................38
4 Axisymmetric Separated Boundary Layer..........................................................41
  4.1 Problem Overview..........................................................................................41
  4.2 Simulation Set up ..........................................................................................41
    4.2.1 Structured Meshes ..................................................................................41
    4.2.2 Analysis Conditions..............................................................................42
  4.3 SST Turbulence Model Results and Conclusions ............................................43
  4.4 SA Turbulence Model Results and Conclusions ............................................52

5 Flow over NACA 0012 Airfoil ........................................................................62
  5.1 Problem Overview..........................................................................................62
  5.2 Structured Meshes .......................................................................................62
  5.3 Analysis Conditions .....................................................................................63
  5.4 SST Turbulence Model Results and Conclusions ..........................................65
    5.4.1 Additional Analysis for SST Turbulence Model ....................................70
  5.5 SA Turbulence Model Results and Conclusions ............................................74
  5.6 Unstructured Meshes ...................................................................................76
    5.6.1 Outer Mesh ............................................................................................78
    5.6.2 Inner Mesh .............................................................................................78
    5.6.3 Merged Mesh ..........................................................................................79
  5.7 Analysis Conditions .....................................................................................80
  5.8 SST Turbulence Model Results and Conclusions ..........................................81
  5.9 VB Application ..............................................................................................89
  5.10 Conclusions of Structured and Unstructured Meshes .................................90

6 Flow over NACA 4412 Airfoil ........................................................................92
  6.1 Problem Overview..........................................................................................92
  6.2 Structured Meshes .......................................................................................92
  6.3 Analysis Conditions .....................................................................................93
  6.4 SST Turbulence Model Results and Conclusions ............................................94
  6.5 SA Turbulence Model Results and Conclusions ............................................100
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.6</td>
<td>Unstructured Meshes</td>
<td>103</td>
</tr>
<tr>
<td>6.7</td>
<td>Analysis Conditions</td>
<td>107</td>
</tr>
<tr>
<td>6.8</td>
<td>SST Turbulence Model Results and Conclusions</td>
<td>107</td>
</tr>
<tr>
<td>6.8.1</td>
<td>Additional Analysis [1]: Further Mesh Refinement and Conclusions</td>
<td>114</td>
</tr>
<tr>
<td>6.8.2</td>
<td>Additional Analysis [2]: Varied Prism Layer Settings and Conclusions</td>
<td>121</td>
</tr>
<tr>
<td>6.8.3</td>
<td>Additional Analysis [3]: Different Turbulence Models and Conclusions</td>
<td>128</td>
</tr>
<tr>
<td>6.9</td>
<td>Comparing Structured and Unstructured Mesh Results</td>
<td>134</td>
</tr>
<tr>
<td>7</td>
<td>Conclusions</td>
<td>136</td>
</tr>
<tr>
<td>8</td>
<td>Reference</td>
<td>138</td>
</tr>
</tbody>
</table>
List of Figures

Figure 2-1 Typical velocity profile for a turbulent boundary layer ......................... 8
Figure 3-1: 2D zero pressure gradient flat plate grid 69 x 49 ................................. 16
Figure 3-2: Flat plate boundary conditions .............................................................. 16
Figure 3-3: Comparison of skin friction coefficient (Cf vs. X) of SST turbulence model ......................................................................................................................... 20
Figure 3-4: Comparison of skin friction coefficient (Cf vs. Re$^\theta$) of SST turbulence model ......................................................................................................................... 23
Figure 3-5: Comparison of boundary layer profiles of SST turbulence model ....... 25
Figure 3-6: Comparison of skin friction coefficient (Cf vs. X) of additional results for SST turbulence model ............................................................................................................. 28
Figure 3-7: Comparison of skin friction coefficient (Cf vs. X) of SA turbulence model ................................................................................................................................. 31
Figure 3-8: Comparison of skin friction coefficient (Cf vs. Re$^\theta$) of SA turbulence model ................................................................................................................................. 34
Figure 3-9: Comparison of boundary layer profiles of SA turbulence model ........ 36
Figure 3-10: Comparison of velocity contour with basic settings: SA (left) vs SST (right) ........................................................................................................................................... 37
Figure 3-11: Comparison of skin friction coefficient (Cf vs. X) of additional results for SA turbulence model .......................................................... 40
Figure 4-1: Axisymmetric separated boundary layer grid 357 x 97 ......................... 42
Figure 4-2: Driver axisymmetric separated boundary conditions ...................... 42
Figure 4-3: Comparison of pressure coefficient on the low wall of SST turbulence model

Figure 4-4: Comparison of velocity profiles on different locations of SST turbulence model

Figure 4-5: Comparison of pressure coefficient on the low wall of SA turbulence model

Figure 4-6: Comparison of velocity profiles on different locations of SA turbulence model

Figure 5-1: Structured mesh grid 449 x 129. The left is the overall view of the mesh. The right is the close-up view of the airfoil

Figure 5-2: Boundary conditions for 0 degree AoA

Figure 5-3: Boundary conditions for 10 and 15 degree AoA

Figure 5-4: Comparison of pressure coefficient on the airfoil surface of SST turbulence model

Figure 5-5: Comparison of pressure coefficient of additional analysis results SST turbulence model

Figure 5-6: Comparison of the convergence status between previous and additional analysis. The left indicates the Mesh 3 and the right indicates the previous Mesh 3

Figure 5-7: Comparison of pressure coefficient on the airfoil surface of SA turbulence model

Figure 5-8: The closed volume of center part

Figure 5-9: The closed volume of outer part

Figure 5-10: The octant levels of outer part
Figure 5-11: Close up of MeshUS4X0 AoA (Note that 6 prism layers on airfoil surface; 1st prism = 0.5mm; growth ratio = 1.1) ................................................................. 79

Figure 5-12: Merged MeshUS4 inner and out parts .............................................. 80

Figure 5-13: Comparison of pressure coefficient on the airfoil surface SST turbulence model ................................................................. 84

Figure 5-14: Comparison of pressure coefficient of results SST and SA turbulence models ........................................................................................................ 85

Figure 5-15: The convergence status of MeshUS4X15 AoA. The left indicates the convergence status and the right indicates the summation of pressure on airfoil ............ 87

Figure 6-1: Structured mesh grid 449 x 129. The left is the overall view of the mesh. The right is the close-up view of the airfoil ................................................................. 93

Figure 6-2: Boundary conditions for 13.87 degree AoA ........................................ 93

Figure 6-3: Comparison of pressure coefficient on the airfoil surface of SST turbulence model ........................................................................................................ 97

Figure 6-4: Comparison of velocity profile on different locations of SST turbulence model ........................................................................................................ 99

Figure 6-5: Comparison of pressure coefficient on the airfoil surface of SA turbulence model ........................................................................................................ 101

Figure 6-6: Comparison of velocity profile on the different locations of SA turbulence model ........................................................................................................ 101

Figure 6-7: Convergence status sample: Mesh S1 .............................................. 102

Figure 6-8: The geometry model of unstructured mesh ....................................... 104

Figure 6-9: Close-up view of the mesh center ...................................................... 104
Figure 6-10: Different meshes around the leading edge: (a) Mesh US4, (b) Mesh US2, (c) Mesh US11, (d) Mesh US0.5 ................................................................. 105

Figure 6-11: Different meshes around the airfoil section: (a) Mesh US11, (b) Mesh US12, (c) Mesh US13 ........................................................................................................... 106

Figure 6-12: Comparison of pressure coefficient on the airfoil surface of SST turbulence model .............................................................................................................. 110

Figure 6-13: Comparison of velocity profiles on different locations of SST turbulence model .................................................................................................................. 113

Figure 6-14: Close-up views of different ranges of octant size for computational meshes in the airfoil region: (a) 1 mm x 20, (b) 1 mm x 10, (c) 1 mm x 4, (d) 1 mm x 2 .... 115

Figure 6-15: Comparison of pressure coefficient on the airfoil surface of additional analysis [1] .................................................................................................................................. 118

Figure 6-16: Comparison of velocity profiles on different locations of additional analysis [1] .................................................................................................................. 119

Figure 6-17: Computational mesh of different parameters of prism elements: (a) Mesh USR1, (b) Mesh USR2, (c) Mesh USR3, (d) Mesh USR4, (e) Mesh USR5 ........................ 122

Figure 6-18: Comparison of pressure coefficient on the airfoil surface of additional analysis [2] .................................................................................................................. 124

Figure 6-19: Comparison of velocity profile on different locations of additional analysis [2] .................................................................................................................. 127

Figure 6-20: Streamlines by different turbulence models (Mesh US11) ............... 129

Figure 6-21: Comparison of pressure coefficient on the airfoil surface of different turbulence models .............................................................................................................. 131
Figure 6-22: Comparison of velocity profiles on different locations of different turbulence models ................................................................................................................. 133
List of Tables

Table 3-1: Cycles to convergence and run times for SST turbulence model .......... 26
Table 3-2: Cycles to convergence and run times for SA turbulence model .......... 37
Table 4-1: Cycles to convergence and run times for SST turbulence model (Note that * means the runs didn’t converge) .......................................................... 51
Table 4-2: Cycles to convergence and run times for SA turbulence model .......... 60
Table 5-1: The thickness of the first and the second prism layer of structured mesh. 63
Table 5-2: Cycles to convergence and run times for SST turbulence model .......... 69
Table 5-3: Comparisons of peak (negative) value of Cp comparisons of peak (negative) value of Cp for SST turbulence model .......................................................... 69
Table 5-4: Cycles to convergence and run times for additional analysis ............ 73
Table 5-5: Comparisons of peak (negative) value of Cp additional test for additional analysis.................................................................................................................. 73
Table 5-6: Cycles to convergence and run times for SA turbulence model .......... 76
Table 5-7: Comparisons of peak (negative) value of Cp for SA turbulence model . 76
Table 5-8: Prism layer parameters, growth ratios are fixed at 1.1. (Note: The motivation for the “US0.125 20P” mesh was to reach a Y+ value as low as the finest structured mesh that converged (449x129)) ......................................................... 79
Table 5-9: Cycles to convergence and run times for unstructured mesh. (Note that * The unusually long run times for US2 and US1 may be due to underperforming compute nodes on the cluster machine)........................................................................ 86
Table 5-10: Comparisons of peak (negative) value of Cp for unstructured mesh..... 87
Table 6-1: Cycles to convergence and run time for SST and SA turbulence model 102

Table 6-2: Comparisons of the peak (negative) value of Cp with CFL3D results ... 102

Table 6-3: Unstructured mesh sizes ........................................................................... 104

Table 6-4: Unstructured mesh sizes and prism information ....................................... 105

Table 6-5: Cycles to convergence and run times for SST turbulence model............. 113

Table 6-6: Comparisons of the peak (negative) value of Cp with CFL3D results for SST turbulence model ........................................................................................................... 113

Table 6-7: Range of octant size of different meshes ...................................................... 114

Table 6-8: Cycles to convergence and run times for additional analysis [1]............. 120

Table 6-9: Comparisons of the peak (negative) value of Cp with CFL3D results for additional analysis [1] ........................................................................................................... 120

Table 6-10: Parameters of prism layer elements ......................................................... 121

Table 6-11: Cycles to convergence and run times for additional analysis [2].......... 127

Table 6-12: Comparisons of the peak (negative) value of Cp with CFL3D results for additional analysis [2] ........................................................................................................... 127

Table 6-13: Cycles to convergence and run times for different turbulence models . 133

Table 6-14: Comparisons of the peak (negative) value of Cp with CFL3D results for different turbulence models. (Note that SST, AKN and SKE results are compared to CFL3D SST results and SA result is compared to CFL3D SA result) ........................................... 133
Nomenclature

C    Chord length (m)
C_p  Pressure coefficient
C_f  Skin friction coefficient
k    Turbulence kinetic energy
ε    Turbulent dissipation
v    Inflow velocity
ω    Specific dissipation rate; vorticity vector magnitude
θ    The momentum thickness
P    Production tensor
P_∞  Pressure in free stream
x    X-direction location
y    Y-direction location
d    Distance from the field point to the nearest wall
u_τ  Friction velocity
τ_ω  Wall shear stress
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>(v)</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>(v_t)</td>
<td>Turbulence kinematic viscosity</td>
</tr>
<tr>
<td>(R_e L)</td>
<td>Reynolds number based on length (L)</td>
</tr>
<tr>
<td>(R_e \theta)</td>
<td>Reynolds number based on momentum thickness</td>
</tr>
<tr>
<td>(Y_M)</td>
<td>Contribution of the fluctuating dilatation</td>
</tr>
<tr>
<td>(G_K)</td>
<td>Generation of turbulence kinetic energy due to mean velocity gradient</td>
</tr>
<tr>
<td>(G_b)</td>
<td>Generation of turbulence kinetic energy due to buoyancy</td>
</tr>
<tr>
<td>(y^+)</td>
<td>Non-dimensional wall normal distance</td>
</tr>
<tr>
<td>(u^+)</td>
<td>The dimensionless velocity</td>
</tr>
<tr>
<td>(y^+)</td>
<td>The wall coordinate</td>
</tr>
<tr>
<td>(\mu)</td>
<td>Molecular dynamic viscosity</td>
</tr>
<tr>
<td>(\mu_t)</td>
<td>Turbulence eddy viscosity</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Fluid density (kg/m(^3))</td>
</tr>
<tr>
<td>(\sigma к, \sigma_\epsilon)</td>
<td>Turbulent prandtl number for (k) and (\epsilon)</td>
</tr>
<tr>
<td>(\Omega)</td>
<td>Specific dissipation rate, vorticity vector magnitude</td>
</tr>
<tr>
<td>(f_\mu)</td>
<td>Damping function</td>
</tr>
<tr>
<td>(\tau_{ij})</td>
<td>Reynolds stress tensor</td>
</tr>
</tbody>
</table>
\( S_{ij} \) Mean strain-rate tensor
\( \delta_{ij} \) Kronecker delta
\( F_1, F_2 \) Blending function
\( \phi_1, \phi_2 \) Closure coefficients
\( \gamma_1, \gamma_2 \) Closure coefficients
\( \sigma_{k1}, \sigma_{k2} \) Closure coefficients
\( \sigma_{\omega1}, \sigma_{\omega2} \) Closure coefficients
\( \beta_1, \beta_2, \beta_* \) Closure coefficients
\( C_1, C_2 \) Closure coefficients
\( C_{b1}, C_{b2} \) Closure coefficients
\( C_{w1}, C_{w2}, C_{w3} \) Closure coefficients
\( C_{v1}, C_{t3}, C_{t4} \) Closure coefficients
\( C_{1E}, C_{2E}, C_{3E} \) Closure coefficients
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>Two-dimensional</td>
</tr>
<tr>
<td>AoA</td>
<td>Angle of Attack</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamic</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>SA</td>
<td>The Spalart-Allmaras</td>
</tr>
<tr>
<td>SKE</td>
<td>Standard k-ε</td>
</tr>
<tr>
<td>AKN</td>
<td>The low-Reynolds-number model, the Abe-Nagano-Kondoh</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

I would never have been able to finish my master degree without the guidance of my master director Dr. George Huang. To be honest, it is very difficult for an international student to adapt to the life of study abroad. The first time I saw Dr. Huang was to talk about the scholarship. From then on, I have decided to work with him. In these two years, I have learned lots of skills from him, especially the attitude for academic work.

Moreover, without Dr. David Welsh’s help, who is working in the Cradle North America Company, I would never have finished my research quickly. When I was working on my projects, he came up with many ideas, which taught me a lot about the SC/Tetra software. Every mission he gave me was a tough task that I had no idea how to complete in that field. So, I had to push myself to study and to finish that as soon as possible. Fortunately, looking back on the past, if there was no tough task, I would never have built up the confidence facing all the problems and solving them.

Likewise, I want to say thanks for my family, for everything they have done for me. Without their suggestions, I would not have the opportunity to study in America. Without their encouragements, I would not regain confidence when I had setbacks. Without their care, I would feel like loneliness and fighting myself.
1 Introduction

1.1 Motivation

Computational fluid dynamics (CFD) has become a huge concern in the past four decades. CFD is the simulation of fluid engineering systems using modeling and numerical methods. It enables scientists and engineers to perform “numerical experiments” (computer simulations) in a “virtual flow laboratory.” Numerical simulation of fluid flow is useful for many fields in our lives. For example, designers of vehicles improve the aerodynamic characteristics; chemical engineers maximize the yield from their equipment; meteorologists forecast the weather and warn of natural disasters.

In this thesis, we are concentrating on geometry and domain, initial and boundary conditions, and different turbulence models for applications. Four fundamental simulation cases were analyzed: a 2D zero pressure gradient flat plate, an axisymmetric separated boundary layer, the NACA 0012 airfoil, and the NACA 4412 airfoil. Structured meshes that are characterized by regular connectivity were used for all cases. Unstructured meshes that are characterized by irregular connectivity were also tested for the two airfoil cases.

Today, more and more computer simulations of physical processes are used in scientific research. Simulations are used for environmental predictions, such as the analysis of surface water processes and the risk assessment of underground nuclear waste repositories. In the medical field, CFD simulations can be used, for example, to predict vessel aneurysms, concentrating on the wall shear stress and pressure on the aneurysmal sac, then comparing results to get characterizations for rupture and non-rupture cases (Cebral et al. (2005)). For
engineered systems, terminology such as “virtual prototyping” and “virtual testing” is now being used in development activities to define numerical simulation for the design, evaluation, and “testing” of new hardware and even entire systems (Oberkampfa & Trucanob (2002)).

Validation is the assessment of the accuracy of a computational simulation by comparison with experimental data or an analytical solution. In this thesis, the goal is to validate the accuracy of the SC/Tetra CFD code. This includes the investigation of different turbulence models, grid convergence, different wall functions for the SST k-ω turbulence model, and structured and unstructured meshes.

Claims and suggestions in the literature that the verification or validation of CFD numerical models has been achieved for fluidized beds have been shown to be inconsistent with objective criteria and the accepted usage of terminology (Grace & Taghipour (2004)). Liangzhu Wang has pointed out that air momentum effects, contaminant concentrations, and air temperatures are uniformly and homogeneously distributed in a zone of a building. A CFD program has been developed to improve the multizone model by applying CFD techniques to the poorly-mixed zones as well as the rest of the zones (Wang & Chen (2007)). There is extensive literature on validation, including policy statements (Oreskes et al. 1994). Papers dealing with the verification and validation of CFD codes also relate to many fields, e.g. aeronautics and aerospace (Oreskes et al 1994).
1.2 Four Analyzed Cases

1.2.1 2D Zero Pressure Gradient Flat Plate Validation

Perhaps one of the most fundamental test cases to perform is the 2D zero pressure gradient flat plate case. This case should be run at a Reynolds number per unit length of Re = 5 million, which is sufficient to achieve the desired Re theta levels based on the free stream velocity and inflow properties carried out. The grids are given in Chapter 3, Figure 3-1, along with typical boundary conditions. For this case the maximum boundary layer thickness is about 0.03 L, so the grid height of y = L is far enough away to have very little influence. In the results, the results of SC/Tetra, CFL3D results (Langley Research Center), and k-s data (Schoenherr (1932)) are compared to validate the accuracy of SC/Tetra CFD code.

1.2.2 Axisymmetric Separated Boundary Layer

The experiment utilized a cylinder of 0.140 m diameter in a tunnel in which an adverse pressure gradient was imposed over a portion of the flow by diverging the four tunnel walls. Each wall was deflected by as much as 0.045 m, resulting in an area expansion ratio of about 1.6. In the experiment, the tunnel sidewall boundary layers were thinned via suction. It is important to note that this axisymmetric case is not a 2-D computation; it uses a periodic (rotated) grid system with appropriate boundary conditions on the periodic sides of the grid. The experimentalist provided a streamline shape well outside of the cylinder's boundary layer that can be used as an inviscid surface for defining the upper boundary condition in a CFD simulation. The inflow to the domain is adjusted so that the naturally developing turbulent boundary layer on the cylinder in the CFD solution grows to approximately 0.012 m thick near
the position \( x = -0.3 \, \text{m} \) (just upstream of the adverse pressure gradient), as noted in the experiment. A short region with symmetry is imposed upstream to avoid possible incompatibilities between free stream inflow and wall BCs. The experimental data and CFL3D results (Driver (1991)) are used to compare with the SC/Tetra results.

1.2.3 Flow over NACA0012 Airfoil

A simple and extensively documented airfoil, the NACA0012, provided an opportunity to compare single phase SC/Tetra results with CFL3D data. For the purposes of this validation, the definition of the NACA 0012 airfoil is slightly altered so that the airfoil closes at chord = 1 with a sharp trailing edge. To do this, the exact NACA 0012 formula is used, and then the airfoil is scaled down by 1.008930411365. The scaled formula can be written as:

\[
y = \pm 0.594689181 \times [0.298222773 \times \sqrt{x} - 0.127125232 \times x - 0.357907906 \times x^2 + 0.291984971 \times x^3 - 0.105174696 \times x^4]
\]

The turbulent NACA 0012 airfoil case should be run at essentially incompressible conditions at Mach number 0.15, and the Reynolds number per chord is \( \text{Re number} 6e+6 \). Boundary layers should be fully turbulent over most of the airfoil. Inflow conditions for the turbulence variables should be reported. To minimize issues associated with effect of the far field boundary, the far field boundary in the grids provided has been located almost 500 chords away from the airfoil for structured meshes. Otherwise, a "far field point vortex" boundary condition correction should be employed (Thomas & Salas (1986)). The sample grid plot is shown in Chapter 5, Figure 5-1. The SC/Tetra result is compared with CFL3D results. The experimental data presented the pressure and drag coefficient (McCroskey (1988)).
1.2.4 Flow over NACA4412 Airfoil

Sang-Tae Chung has demonstrated the simulation of the NACA4412 airfoil based on different turbulence models on pressure coefficient and velocity profiles (Chung et al. (1998)). The 2D experiment developed the NACA4412 airfoil. The definition of the airfoil shape is slightly altered so that the airfoil closes at chord = 1 with a sharp trailing edge. Flow field characteristics were measured with a flying hot-wire for the airfoil at a 13.87 degree angle of attack. The Reynolds number was 1.52 million per airfoil chord. Also, note that the CFD is performed here on grids with a far field outer boundary extending to 100 c. Moreover, the sample grid is shown in Chapter 6, Figure 6-1. The experimental data for this case is provided at thousands of locations in the field surrounding the trailing edge region of the airfoil (Coles & Wadcock (1979)) (Wadcock (1979)).
2 Methodology

2.1 Introduction to the SC/Tetra Software

In this work, all cases were run using Cradle SC/Tetra software, which was developed by Software Cradle Limited Company in 1984. The Cradle SC/Tetra v10 was used in this work. This software is more powerful based on these five characteristics: user friendly interface and usability; powerful mesher; low memory consumption; outstanding computation speed and accuracy; and state of the art postprocessor. The SC/Tetra CFD code consists of three parts: SC/T preprocessing, SC/T solver, and SC/T postprocessing. First, in the SC/T preprocessing, it is easy for the user to import a model file from AUTOCAD or Solid works and define surfaces and closed volume. Then, creating grids is made simpler by the octree function. It not only refines the entire model, but also allows local refinement for significant corners and meshes the model with different prism layers inserted. For example, the outer side domain, which the analysis shows does not impact much on the results, can be given a bigger octant size, and the major part can be defined as a smaller size. The analysis condition is easily applied, including flow type; different turbulence models; boundary conditions; etc. Furthermore, the case would be calculated by the SC/T solver that plays a role in handling the calculation and monitoring the progress and convergence status. In the screen, the user is able to check all variables values in the L file from the tool bar. Once the convergence status is starting to diverge or appearing to be a huge value, the user can stop running immediately. Additionally, simulations are performed on two kinds of machines. The Hardware #1 is a Lenovo laptop with Windows 8 Professional X64, and the CPU is Intel ® Core™ i7-3630QM (quad-core) @ 2.40 GHz with 16 GB RAM. The Hardware #2 is called Cradle-Cluster with Fedora (Linux)
Release 11 (64-bit); it has 47 compute nodes, and each node has a 3.4GHz AMD Phenom II X4 965 (quad-core) for CPU and 8GB RAM. All simulations were run using 4 nodes, to give 16-parallel. Last but not least, the SC/T postprocessing deals with the results. For example, with pressure contour, velocity profiles by drawing several lines and generating an animation.

2.2 The Law of the Wall

In fluid dynamics, the law of the wall claims that the average velocity of a turbulent flow at a certain point is proportional to the logarithm of the distance from that point to the "wall," or the boundary of the fluid region. This law of the wall was first published by Theodore von Kármán in 1930 (von Kármán et al. (1930)). It is only technically applicable to parts of the flow that are close to the wall (<20% of the height of the flow), though it is a good approximation for the entire velocity profile of natural streams. Three different layers constitute the flow near the wall region. The inner layer is very close to the wall and is controlled by viscous stress, and the outer layer is far from the wall and is dominated by momentum transport due to Reynolds stresses. Finally, the last layer can be found in between, transferring the inner layer to the outer layer.

A velocity profile for a turbulent boundary layer is illustrated in Figure 2-1, where $y^+$ is the non-dimensional wall normal distance defined as:

$$y^+ = \frac{y u_\tau}{v} \quad (2.1)$$

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (2.2)$$
where $u_\tau$ is the friction velocity, $\nu$ is the kinematic viscosity, $\tau_w$ is the wall shear stress and $\rho$ is density.

\[ u^+ = \frac{u}{u^*}, \quad y^+ = \frac{y^*}{\nu}, \quad u^* = \sqrt{\frac{\tau_w}{\rho}} \]

\[ u^+ = \frac{1}{\kappa} \ln y^+ + C \]

Figure 2-1 Typical velocity profile for a turbulent boundary layer

Different turbulent models claim that the first layer of computational cells are either in the log layer or in the viscous layer where $u^+ = y^+$. Since the $y^+$ values are dependent on flow characteristics, they are not available during pre-processing. Therefore, analyzing the $y^+$ values during post-processing plays an important role in controlling their correspondence with the needs of the turbulent model. To use these equilibrium profiles effectively, it is desirable that the grid spacing be such that the near-wall node lies within the logarithmic layer (Apsley (2007)). If separation is important, integrating the domain to the wall is possible by using a low-Reynolds number model, but one must make sure one has sufficient points to cover the
boundary layers ($y^+<1$). It is possible to use a low Reynolds number model with an adaptive wall function technique, although this may not be that useful. Prism layers near the wall are needed in all cases to ensure numerical accuracy when building the mesh. Two rules are used for inserting prism layers: 2-3 layers when using the standard wall function; 10-30 layers when using low Reynolds number models.

2.3 Turbulence Models

Turbulence modeling is a crucial topic in most CFD simulations. Virtually all engineering applications are turbulent and hence require a turbulence model. Several types of turbulent flow constitute this issue. First, Direct Numerical Simulation (DNS) of turbulent flows for practical applications is not possible. Second, Large Eddy Simulation (LES) or Detached Eddy Simulation (DES) are only limited for low Reynolds number flow. Last, Reynolds-Averaged Navier-Stokes (RANS) methods are probably the most popular approach for solving practical fluid flow problems. Four turbulence models are introduced below, which are tested for simulations in this work. Three of them are two-equation models (SST, SKE, and AKN), and SA is a one-equation model. Two-equation turbulence models have been used for almost fifty years. Most of these models solve a transport equation for turbulence kinetic energy ($k$) and a second transport equation solves for turbulent dissipation ($\varepsilon$) or turbulent specific dissipation ($\omega$).

2.3.1 The Shear-Stress Transport (SST) $k-\omega$ Turbulence Model

The SST $k-\omega$ turbulence model is a two equation eddy-viscosity model which is more accurate and much more numerically stable in the near wall region. The use of a $k-\omega$
formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer; hence the SST k-\( \omega \) model can be used as a Low-Reynolds turbulence model without an extra damping function. For the two equations, the first is called Turbulence Kinetic Energy, and the second is called the Specific Dissipation Rate. The two equations (Menter (1994)) are:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right],
\]

(2.1)

and

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_i} = \frac{\gamma}{\nu_t} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho a_t \omega^2}{\rho} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} + 2(1 - F_1) \frac{\rho a_t \omega^2}{\rho} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}
\]

(2.2)

where

\[
P = \tau_{ij} \frac{\partial u_i}{\partial x_i}
\]

(2.3)

\[
\tau_{ij} = \mu_t \left( 2S_{ij} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij}
\]

(2.4)

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

(2.5)

and the turbulence eddy viscosity is computed from:

\[
\mu_t = \frac{\rho a_t k}{\max(a_t, \omega, a_t F_2)}
\]

(2.6)

Each of the constants is a blend of an inner (1) and outer (2) constant, blended via:

\[
\phi = F_1 \phi_1 + (1 - F_1) \phi_2
\]

(2.7)

where the \( \phi_1 \) represents the coefficient from the k-\( \omega \) model and \( \phi_2 \) represents the coefficients of the k-\( \epsilon \) model. The blending function \( F_1 \) is given by:
\[ F_1 = \tanh(\arg_1^4) \]  
\hspace{0.5cm} (2.7)

where

\[ \arg_1 = \min \left[ \max \left( \frac{\sqrt{K}}{\beta^\omega d}, \frac{500\nu}{a^2}, \frac{4\rho \sigma_{\omega{k}^2}}{CD_{k\omega} a^2} \right) \right] \]  
\hspace{0.5cm} (2.8)

where \( F_2 \) is given by:

\[ F_2 = \tanh(\arg_2^2) \]  
\hspace{0.5cm} (2.9)

where

\[ \arg_2 = \max \left( 2 \frac{\sqrt{K}}{\beta^\omega d}, \frac{500\nu}{a^2} \right) \]  
\hspace{0.5cm} (2.10)

Here \( \rho \) is the density, \( \nu_t = \mu_t / \rho \) is the turbulent kinematic viscosity, \( \mu \) is the molecular dynamic viscosity; \( d \) is the distance from the field point to the nearest wall; \( \Omega \) is the vorticity magnitude; and \( CD_{k\omega} \) is the positive portion of the cross-diffusion term. The model constants are:

\[ \gamma_1 = \frac{\beta_1}{\beta^*}, \gamma_2 = \frac{\beta_2}{\beta^*}, \sigma_{k1} = 0.5, \sigma_{k2} = 1.0, \sigma_{\omega1} = 0.5, \sigma_{\omega2} = 0.856, \beta_1 = 0.075, \beta_2 = 0.0828, \beta^* = 0.09, k=0.41, \alpha_1 = 0.31 \]  
\hspace{0.5cm} (2.11)

2.3.2 The Spalart-Allmaras (SA) Turbulence Model

The SA turbulence model is widely used in the aviation industry, and it is the one-equation model (Spalart (1994, 2000, and 2007)) in which the equation of eddy viscosity is directly solved. The goal was to produce a turbulent transport model that was fast, numerically stable, and reasonably accurate for both the shear layer and boundary layers. The model uses
a turbulence variable $\tilde{\vartheta}$ that is identical to the turbulent kinematic viscosity in the near-wall region. The one-equation model is given by the following equation:

$$\frac{\partial \tilde{\vartheta}}{\partial t} + u_j \frac{\partial \tilde{\vartheta}}{\partial x_j} = \tilde{c}_{b1}(1 - f_{t2})\dot{S}\tilde{\vartheta} - \left[ c_{w1} f_w - \frac{c_{b1}}{k^2} f_{t2} \right] \left( \frac{\tilde{\vartheta}}{d} \right)^2 + \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( (\nu + \tilde{\vartheta}) \frac{\partial \tilde{\vartheta}}{\partial x_j} \right) + c_{b2} \frac{\partial \tilde{\vartheta}}{\partial x_i} \frac{\partial \tilde{\vartheta}}{\partial x_i} \right]$$

(2.12)

Here, the first part of the right hand side is the production term; the second part is the dissipation, and

$$f_{t2} = c_{t3} \exp(-c_{t4} x^2) \quad \text{(2.13)}$$

$$\dot{S} = \Omega + \frac{\tilde{\vartheta}}{k^2 d^2} f_{v2} \quad \text{(2.14)}$$

Here $\Omega$ is the magnitude of the vorticity, and $d$ is the distance from the field point to the nearest wall.

$$f_{v2} = 1 - \frac{x}{1+x f_{v1}} \quad \text{(2.15)}$$

$$f_w = g \left[ \frac{1+c_{w2}}{g_6+c_{w3}} \right]^{1/6} \quad \text{(2.16)}$$

and the remaining functions are given by:

$$f_{v1} = \frac{x^3}{x^3 + c_{v1}^3} \quad \text{(2.17)}$$

$$g = r + c_{w2} (r^6 - r) \quad \text{(2.18)}$$

$$r = \min \left[ \frac{\tilde{\vartheta}}{5 k^2 d^2}, 10 \right] \quad \text{(2.19)}$$

The turbulence eddy viscosity is computed from:

$$\mu_t = \rho \tilde{\vartheta} f_{v1} \quad \text{(2.20)}$$
The constants are:

\[ c_{b1} = 0.1355, c_{b2} = 0.622, c_{w1} = \frac{c_{b1}}{k^2} + \frac{1+c_{b2}}{\sigma}, c_{w2} = 0.3, c_{w3} = 2, c_{v1} = 7.1, k = 0.41, \sigma = 2/3, c_{\ell3} = 1.2, c_{\ell4} = 0.5. \]  

(2.21)

2.3.3 The Standard k-\( \epsilon \) Turbulence Model

The standard k-\( \epsilon \) model is also a two-equation model that is used for fluid dynamics problems. The turbulence kinetic energy, k, and its rate of dissipation, \( \epsilon \), are obtained from the following transport equations:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_k}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k
\]

(2.22)

and

\[
\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_k}{\sigma_k}) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon
\]

(2.23)

In the above equations, \( G_k \) is the generation of turbulence kinetic energy. \( G_b \) is the generation of turbulence kinetic energy due to buoyancy. \( Y_M \) is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. \( C_{1\epsilon}, C_{2\epsilon}, C_{3\epsilon} \) are constants. \( \sigma_k \) and \( \sigma_{\epsilon} \) are the turbulent Prandtl numbers for k and \( \epsilon \), respectively.

2.3.4 The Abe-Nagano-Kondoh (AKN) Turbulence Model

The AKN turbulence model (Abe & Kondoh (1992)) has three characteristics. First, it allows accurate analysis of a wide range of flows, with Reynolds numbers ranging from low to high. Second, it plays an important role in conventional turbulence models in terms of accuracy of prediction of separation and reattachment flows. Third, it is also valid for analysis
of the transition from laminar flow to turbulent flow, or the reverse transition from turbulent flow to laminar flow. Recently, this model has also been used as a basis for the proposal of turbulence models that allow analysis of thermal stratification, and it is likely to find application in a wide range of fields (Murakami et al. (1996)). First, the eddy viscosity is:

$$\mu_t = \rho C_t f_\mu \frac{K^2}{\varepsilon}$$

(2.24)

where $f_\mu$ is the damping function, and the equations for turbulence energy $k$ and dissipation rate $\varepsilon$ are as shown below:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial u_i \rho k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_i} \right] + G_s + G_T - \rho \varepsilon$$

(2.25)

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial u_i \rho \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_1 \frac{\varepsilon}{k} (G_s + G_T) - C_2 f_2 \frac{\rho \varepsilon^2}{k}$$

(2.26)

Here, in the region very close to the wall, the final term on the right-hand side of the $\varepsilon$ equation would be subject to divergence because $k$ tends to be zero. The model functions and constants are:

$$f_\mu = \left\{ 1 - \exp \left( -\frac{y^*}{14} \right) \right\}^2 \left[ 1 + \frac{5}{R_t^3} \exp \left\{ -\left( \frac{R_t}{200} \right)^2 \right\} \right]$$

(2.27)

$$f_2 = \left\{ 1 - \exp \left( -\left( \frac{y^*}{3.1} \right) \right) \right\}^2 \left[ 1 - 0.3 \exp \left\{ -\left( \frac{R_t}{6.5} \right)^2 \right\} \right]$$

(2.28)

where $y^* = \frac{u_e y}{v}, \ u_e = (v \varepsilon)^{0.25}, R_t = \frac{K^2}{\nu \varepsilon}.$ And, $\sigma_K = \sigma_\varepsilon = 1.4, C_1 = 1.5,$ $C_2 = 1.9, C_t = 0.09.$
3 2D Zero Pressure Gradient Flat Plate

3.1 Problem Overview

The 2D Zero pressure gradient flat plate is validated in this chapter. We analyzed structured mesh and compared the results of skin friction coefficient and velocity profiles with the CFL3D and K-S results. In this chapter, we focus on two turbulence models, the SST and SA turbulence models.

3.2 Simulation Set up

3.2.1 Structured Meshes

The family of five structured meshes is shown below; the sample size is a 69 x 49 grid, which was downloaded from the NASA turbulence modeling resource website, and the grid of the mesh is from coarse mesh (32 x 25) to fine mesh (545 x 385), named “Mesh 1” to “Mesh 5.” From Figure 3-2, the left vertical surface is registered as the “Inlet” face, and the top surface (Y=1) and symmetric surface are registered as the “Top_wall” face. Also, the bottom surface (Y=0, X = 0 to 2) is the “Bottom_wall” face. Finally, the right vertical surface is the “Outlet” face.
Figure 3-1: 2D zero pressure gradient flat plate grid 69 x 49

Figure 3-2: Flat plate boundary conditions
3.2.2 Analysis Conditions

The 2D simulation is a steady state simulation using the Reynolds number $5e+6$, as shown in the Figure 3-2 with a maximum 20,000 cycles. The inflow velocity is set at 1 m/s. (Note that the turbulence properties are applied $6e-07$ and $1e-04$ for $k$ and $e$ only for the SA turbulence model and that the SA model's turbulent kinematic viscosity BC is being modified based on the NASA site guidance.) The boundary condition at the outlet face is set to be the static pressure. The bottom_wall is the no-slip and smooth wall boundary condition. The top_wall is the free-slip wall. Moreover, the selected flow is incompressible flow with density changed to $1 \text{ kg/m}^3$ and viscosity to $2e-07 \text{ kg/(m.s)}$ in order to match the Re number. The convergence criterion is set to $1e-06$, and the under-relaxation coefficient reduces to 0.3 for all values (only for Mesh 1 SA turbulence model because it cannot converge). The SST and SA turbulence model are selected to be tested. Definitions are given here for the relevant quantities, including Re_theta, Cf, $u^+$, and $y^+$:

$$R_{e\theta} = \frac{\rho_\infty U_\infty \theta}{\mu_\infty}$$  \hspace{1cm} (3.2.1)

$$\theta = \int_0^\infty \frac{\rho}{\rho_\infty} \frac{u}{U_\infty} (1 - \frac{u}{U_\infty}) dy$$  \hspace{1cm} (3.2.2)

$$C_f = \frac{\tau_\omega}{2 \rho_\infty U_\infty^2}$$  \hspace{1cm} (3.2.3)

$$u^+ = \frac{u}{v^*} = \frac{u}{\sqrt{\tau_\omega / \rho_\omega}}$$  \hspace{1cm} (3.2.4)

$$y^+ = \frac{y v^*}{\gamma_\omega} = \frac{y \sqrt{\tau_\omega / \rho_\omega}}{\gamma_\omega}$$  \hspace{1cm} (3.2.5)

where $\theta$ is the momentum thickness, $C_f$ is the dissipation of the skin friction coefficient, and $R_{e\theta}$ is the Reynolds number based on the momentum thickness.
3.3 SST Turbulence Model Results and Conclusions

Comparisons are made using the SST Turbulence Model. In addition, the command “SSTD LORE 0” was tested by using with the SST turbulence model, which turns off the Wilcox Low-Reynolds-Number correction. The command is added manually to the S-file. Therefore, “CF_SC/T_Y” indicates that the SST model with the command “SSTD LORE 0” and “CF_SC/T_N” indicates the SST model without the command. All the results are compared with the CFL3D results (NASA CFL3D code, using the SST model and equivalent of “LORE 0,” with the finest mesh (“Mesh 5”)), and k-s results (Flat plate skin friction coefficient as a function of Re-theta, from the Karman-Schoenherr formula variables="Re_theta","Cf"). Evaluations focus on the flat plate skin friction profiles, which focus on the wall skin friction coefficient (Cf vs. X) and (Cf vs. Re_theta), and velocity profiles, which concentrate on U+ vs. y+ (at Reθ= 10,000)

(a) Mesh 1
Figure 3-3: Comparison of skin friction coefficient (Cf vs. X) of SST turbulence model

(d) Mesh 4

(e) Mesh 5
(a) Mesh 1

(b) Mesh 2
(c) Mesh 3

(d) Mesh 4
Figure 3-4: Comparison of skin friction coefficient (Cf vs. Reθ) of SST turbulence model

(a) Mesh 1 (Note: Velocity profiles are for the downstream location where Reθ = 10,000)

(e) Mesh 5
(b) Mesh 2

(c) Mesh 3
Figure 3-5: Comparison of boundary layer profiles of SST turbulence model

(d) Mesh 4

(e) Mesh 5
Table 3-1: Cycles to convergence and run times for SST turbulence model

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, hr:min:sec</th>
<th>Run Platform</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh1 Y</td>
<td>816</td>
<td>398</td>
<td>00:00:33</td>
<td>1</td>
</tr>
<tr>
<td>Mesh1 N</td>
<td>816</td>
<td>1,392</td>
<td>00:00:29</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2 Y</td>
<td>3,246</td>
<td>462</td>
<td>00:01:12</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2 N</td>
<td>3,246</td>
<td>477</td>
<td>00:00:27</td>
<td>1</td>
</tr>
<tr>
<td>Mesh3 Y</td>
<td>13,056</td>
<td>730</td>
<td>00:02:25</td>
<td>1</td>
</tr>
<tr>
<td>Mesh3 N</td>
<td>13,056</td>
<td>1,878</td>
<td>00:05:03</td>
<td>1</td>
</tr>
<tr>
<td>Mesh4 Y</td>
<td>52,224</td>
<td>2,345</td>
<td>00:17:04</td>
<td>2</td>
</tr>
<tr>
<td>Mesh4 N</td>
<td>52,224</td>
<td>29,348</td>
<td>02:34:45</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5 Y</td>
<td>208,896</td>
<td>7,090</td>
<td>04:03:03</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5 N</td>
<td>208,896</td>
<td>39,981</td>
<td>14:37:03</td>
<td>2</td>
</tr>
</tbody>
</table>

In this case, the skin friction coefficient and velocity profiles were considered. From the results of Cf vs X, all meshes agree with the CFL3D data except SC/T_N of Mesh 5, which has a big transition problem. We tried to fix the problem using additional analysis. In another kind of plot, Cf vs Re_theta, SC/T_N is closer to the CFL3D data than SC/T_Y for Mesh 1 and Mesh 2. However, for Mesh 3, the SC/T_N is a little higher so that it is close to the K-S relation, and SC/T_Y is the same as the CFL3D data. Likewise, SC/T_N for Mesh 4 and Mesh 5 are close to the K-S data, but because of the transition problem, the Mesh 5 SC/T_N plot only goes up to Re_theta of around 11,500. SC/T_Y of Mesh 4 and Mesh 5 are the same as the CFL3D data. Moreover, SC/T_N and SC/T_Y of Mesh 1 to Mesh 3 have almost the same convergence cycle and running time. Nevertheless, SC/T_N of Mesh 4 and Mesh 5 got 29,345 and 39,981 convergence cycles, respectively. That is much bigger than SC/T_Y, as are the running times. All velocity profiles are in excellent agreement with the CFL3D and K-S data. Overall, based on the transition problem, convergence cycles, and running times, the results of the runs adding the command “SSTD LORE 0” are much better than the runs without adding the command. Grid convergence was also shown by the results of SC/T_Y.
3.3.1 Additional Results: Changing Inflow Turbulence Properties to Fix the Transition Problem for the SST Turbulence Model

The original SC/T boundary conditions didn’t match the NASA site’s Mach number or turbulence values for the inflow condition. So, Matching the Mach number was tested in the hope that these changes would fix some of the problems with the original results. The following plots are for analysis conditions matching the Mach number and applying turbulence intensity. Only Mesh 5 is used, since it had the transition problem. We define “SC/T_Intensity,” which means applying turbulence intensity conditions at the inlet (turbulence intensity = 0.039% and turbulent viscosity ratio =0.009), “SC/T_Mach,” which means matching Mach number (M = 0.2) of the inlet flow (U = 68.4 m/s, density = 1 kg/m3, viscosity = 1.368e-005 kg/(m.s) ), and “SC/T_Mach&Intensity,” which means matching Mach number and applying turbulence intensity conditions at the inlet. The “SSTD LORE 0” command is not applied to the three new runs, since the transition problem occurred without it, in SC/T_N.
Figure 3-6: Comparison of skin friction coefficient (Cf vs. X) of additional results for SST turbulence model

For additional results, unfortunately, the three methods trying to fix the transition problem of Mesh 5 (no “SSTD LORE 0”) were unsuccessful. Obviously, adjusting the Mach number and turbulence intensity are not the solution to the problem. Furthermore, all of them got much lower values than the CFL3D results, even worse than SC/T_N. Therefore, it is necessary to add the command “SSTD LORE 0” for the SST turbulence model.

3.4 SA Turbulence Model Results and Conclusions

On the contrary, in this part, comparisons are made using the SA Turbulence Model. The command “SASW 1” was tested for the SA turbulence model. This command deploys the standard version of SA. The default version in SC/Tetra is “SA-fv3.” Similarly, “CF_SC/T_Y” is defined as the SA with the command, and “CF_SC/T_N” is defined as the SA without the
command. CFL3D_Cf is NASA CFL3D code, using the standard SA model, (equivalent of “SASW 1”), with the finest mesh (“Mesh 5”).
(b) Mesh 2

(c) Mesh 3
Figure 3-7: Comparison of skin friction coefficient (Cf vs. X) of SA turbulence model.

(d) Mesh 4

(e) Mesh 5
(a) Mesh 1

(b) Mesh 2
(c) Mesh 3

(d) Mesh 4
Figure 3-8: Comparison of skin friction coefficient ($C_f$ vs. $Re_\theta$) of SA turbulence model

(e) Mesh 5

(a) Mesh 1
Figure 3-9: Comparison of boundary layer profiles of SA turbulence model
Table 3-2: Cycles to convergence and run times for SA turbulence model

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, hr:min:sec</th>
<th>Run Platform</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh1_Y</td>
<td>816</td>
<td>6,662</td>
<td>00:00:33</td>
<td>1</td>
</tr>
<tr>
<td>Mesh1_N</td>
<td>816</td>
<td>7,038</td>
<td>00:00:29</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2_Y</td>
<td>3,246</td>
<td>5,112</td>
<td>00:01:12</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2_N</td>
<td>3,246</td>
<td>5,601</td>
<td>00:00:27</td>
<td>1</td>
</tr>
<tr>
<td>Mesh3_Y</td>
<td>13,056</td>
<td>5,178</td>
<td>00:02:25</td>
<td>1</td>
</tr>
<tr>
<td>Mesh3_N</td>
<td>13,056</td>
<td>7,768</td>
<td>00:05:03</td>
<td>1</td>
</tr>
<tr>
<td>Mesh4_Y</td>
<td>52,224</td>
<td>5,104</td>
<td>00:17:04</td>
<td>2</td>
</tr>
<tr>
<td>Mesh4_N</td>
<td>52,224</td>
<td>2,443</td>
<td>02:34:45</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5_Y</td>
<td>208,896</td>
<td>7,637</td>
<td>04:03:03</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5_N</td>
<td>208,896</td>
<td>7,345</td>
<td>14:37:03</td>
<td>2</td>
</tr>
</tbody>
</table>

In order to figure out the transition problems, the velocity contours were compared between SA and SST turbulence models, because the SST turbulence model shows better results.

![Figure 3-10: Comparison of velocity contour with basic settings: SA (left) vs SST (right).](image)

The command “SASW 1” returns the SA turbulence model to its standard version. The Cf vs X plots show that Mesh 1 to Mesh 3 have almost the same results, which have transition problems. However, the results of SC/T_Y of Mesh 4 and Mesh 5 remove the transition
problem, whereas SC/T_N of Mesh 4 and Mesh 5 do not. From fig. 3-11, the SA turbulence model generates more boundary layers than the SST turbulence model so that the thickness of the prism layer of the coarse mesh is not thinner than the boundary layers. Therefore, the finer mesh removes the transition problems. In additional analysis, several methods were tried to fix the transition problem. As the Cf vs Re_theta plots show, the results of Mesh 1 and Mesh 2 have large errors with respect to CFL3D, but the SC/T_N values are a little better than SC/T_Y. SC/T_N of Mesh 3 is close to the K-S data, but SC/T_Y is close to the CFL3D data. For Meshes 4 and 5, both cases give results the same as CFL3D. All velocity profiles are in excellent agreement with the CFL3D and K-S data. The convergence cycle and running time are very close between the SC/T_N and SC/T_Y runs, so the elimination of transition in the finer meshes is the main benefit for using SC/T_Y.

3.4.1 Additional Results: Changing Inflow Turbulence Properties to Fix the Transition Problem for the SA Turbulence Model

The original SC/T boundary conditions didn’t match the NASA site’s Mach number (M = 0.2) or turbulence condition at the inlet (ν_t = 3 * ν). The following plots are for analysis conditions matching one or both of these, as well as higher incoming turbulence levels. The aim was to fix the transition problem for the SA turbulence model. Therefore, Mesh 1 was used because it has the least elements, so that running times are low and the transition problem could be checked quickly. We defined “SC/T_OldBC_#*nu,” which is the SA model’s ν_t = ( # * ν ), “SC/T_Mach_#*nu,” which means Matching Mach number, with ν_t = ( # * ν), and “SC/T_Mach_3*nu_Pseudo-timesteps,” which means Matching Mach number, with ν_t = ( #
* v), and Pseudo-timestep-relaxation factor set to 5 for U, V, W. The command “SASW 1” is applied to all new cases.
In the additional results, all new runs gave the same results as the original. The matched Mach number and high values of nu_turbulent BC cannot improve the results and fix the transition problem. Perhaps the main problem is an issue with the turbulence model.

Figure 3-11: Comparison of skin friction coefficient (Cf vs. X) of additional results for SA turbulence model
4 Axisymmetric Separated Boundary Layer

4.1 Problem Overview

This chapter covers the flow going through the axisymmetric separated boundary layer section. The pressure coefficient and velocity profiles are the key points of this chapter. Five kinds of structured mesh and two turbulence models receive special focus. The results of SC/Tetra, CFL3D, and experimental results are compared.

4.2 Simulation Set up

4.2.1 Structured Meshes

A family of five structured meshes is taken from the NASA Turbulence Modeling resource website, and one kind of mesh is shown below. Different zones have different mesh sizes. The coarsest mesh, 90 x 25, is named “Mesh 1”; likewise, the finest mesh is named “Mesh 5.” All CFL3D and experimental data is provided from validated NASA CFD codes. From Figure 4-2, the left and right surfaces were registered as “Inlet” and “Outlet” faces, respectively. The top and bottom surfaces (x from -1.1 to 1.5) were registered as “Top_wall” and “Bottom_wall” faces, respectively.
4.2.2 Analysis Conditions

Figure 4-1: Axisymmetric separated boundary layer grid 357 x 97

Figure 4-2: Driver axisymmetric separated boundary conditions
The simulation is a steady state simulation using the same Reynolds number $2 \times 10^6$, as shown from the Figure 2-2, with a maximum 50,000 cycles. The inflow velocity is set at 30.2 m/s with the turbulence properties $1 \times 10^{-4}$ for k and e. The boundary condition at the outlet face is set to be the static pressure. The bottom_wall is the no-slip and smooth wall boundary condition. The top_wall is the free-slip wall. Moreover, the selected flow is incompressible flow with density changed to 1 kg/m3 and viscosity to $1.51 \times 10^{-5}$ kg/(m.s) in order to match the Re number and Mach number 0.088 used in the NASA case. The convergence criterion is set to $1 \times 10^{-6}$. The SST and SA turbulence models were used.

Furthermore, the runs without UR (Under-relaxation) used less time and convergence cycles than the runs with UR. Therefore, if the case can converge anyway, it is better not to use UR. If the case cannot converge, however, we should use UR to prevent the code from diverging and make it easier to reach the convergence criteria.

4.3 SST Turbulence Model Results and Conclusions

Comparisons are made using the SST Turbulence Model. As in Chapter 3, “CF_SC/T_Y” and “CF_SC/T_N” were named to reflect whether the command is applied or not. Evaluations focus on the pressure coefficient profile (Low_wall) and velocity profiles (seven locations). From the NASA website: Cp has been shifted so that it is close to 0 near $x=-0.4$ m. Here are the locations of the seven velocity profiles: $X=-0.3302\ m$, $-0.0762\ m$, $0.0508\ m$, $0.1016\ m$, $0.1524\ m$, $0.2286\ m$, and $0.3048\ m$. Cp_Exp is the experimental data, and Cp_CFL3D(SST) = NASA CFL3D code, using the SST model and equivalent of “LORE 0,” with the second finest mesh (713 x 193).
(c) Mesh 3

(d) Mesh 4
Figure 4-3: Comparison of pressure coefficient on the low wall of SST turbulence model

(e) Mesh 5

(a) Mesh 1
(b) Mesh 2

(c) Mesh 3
(d) Mesh 4

(e) Mesh 5
(f) Mesh 1

(g) Mesh 2
Mesh 3

Mesh 4
Figure 4-4: Comparison of velocity profiles on different locations of SST turbulence model

Table 4-1: Cycles to convergence and run times for SST turbulence model (Note that * means the runs didn’t converge)

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, hr:min:sec</th>
<th>Run Platform</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh1_Y</td>
<td>2,136</td>
<td>464</td>
<td>00:00:31</td>
<td>1</td>
</tr>
<tr>
<td>Mesh1_N</td>
<td>2,136</td>
<td>2,000*</td>
<td>00:01:21</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2_Y</td>
<td>8,544</td>
<td>672</td>
<td>00:01:24</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2_N</td>
<td>8,544</td>
<td>2,000*</td>
<td>00:03:22</td>
<td>1</td>
</tr>
<tr>
<td>Mesh3_Y</td>
<td>34,176</td>
<td>1,989</td>
<td>00:08:19</td>
<td>2</td>
</tr>
<tr>
<td>Mesh3_N</td>
<td>34,176</td>
<td>4,000*</td>
<td>00:11:03</td>
<td>2</td>
</tr>
<tr>
<td>Mesh4_Y</td>
<td>136,704</td>
<td>5,878</td>
<td>01:30:16</td>
<td>2</td>
</tr>
<tr>
<td>Mesh4_N</td>
<td>136,704</td>
<td>10,000*</td>
<td>02:26:29</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5_Y</td>
<td>546,816</td>
<td>19,684</td>
<td>24:38:58</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5_N</td>
<td>546,816</td>
<td>25,000*</td>
<td>28:09:18</td>
<td>2</td>
</tr>
</tbody>
</table>

The results of SST with the command “SSTD LORE 0” are in excellent agreement with the CFL3D results, but not with experimental data. However, the results of SST without “SSTD
LORE 0” are very poor. All five meshes did not converge, so the results were obtained by averaging the oscillating values. To fix the problem, several methods were used to improve convergence and the results, but all were unsuccessful. The methods include using under-relaxation: 0.5 for U, V, and W; using pseudo-timestep relaxation: 5 for U, V and W; using matrix solver changes which P: max. Iterations = 1000; solver = AMGCG-STAB; convergence target = 1e-10 and other variables: max. Iterations = 100; solver = MILUCG-STAB; convergence target = 1e-10. Therefore, with all other analysis conditions the same, adding the command “SSTD LORE 0” is a very significant factor to get the runs to converge and give good results.

Furthermore, the CFL3D turbulence settings (0.088% for turbulence intensity and 0.009 for turbulent viscosity ratio) for Mesh 4 were applied to test the difference. The result was that both runs give the same results and are in excellent agreement with the CFL3D results. This confirms that the SC/Tetra results in this case are not overly sensitive to the inflow turbulence settings.

4.4 SA Turbulence Model Results and Conclusions

These comparisons are using the SA Turbulence Model. Similarly, CF_SC/T_Y and CF_SC/T_N are on behalf of the runs with and without the command. The CFL3D data Cp_CFL3D (SA) is from NASA CFL3D code, using the standard SA model (equivalent of “SASW 1”), with the second finest mesh (713x193).
(a) Mesh 1

(b) Mesh 2
(c) Mesh 3

(d) Mesh 4
Figure 4-5: Comparison of pressure coefficient on the low wall of SA turbulence model
(b) Mesh 2

(c) Mesh 3
(d) Mesh 4

(e) Mesh 5
(f) Mesh 1

(g) Mesh 2
(h) Mesh 3

(i) Mesh 4
Figure 4-6: Comparison of velocity profiles on different locations of SA turbulence model

Note that Mesh# with command “SASW 1” is named Mesh#_Y and Mesh# without command “SASW 1” is named Mesh#_N.

Table 4-2: Cycles to convergence and run times for SA turbulence model

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, hr:min:sec</th>
<th>Run Platform</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh1_Y</td>
<td>2,136</td>
<td>7,817</td>
<td>00:03:53</td>
<td>1</td>
</tr>
<tr>
<td>Mesh1_N</td>
<td>2,136</td>
<td>7,787</td>
<td>00:03:49</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2_Y</td>
<td>8,544</td>
<td>8,441</td>
<td>00:15:59</td>
<td>1</td>
</tr>
<tr>
<td>Mesh2_N</td>
<td>8,544</td>
<td>8,532</td>
<td>00:11:52</td>
<td>1</td>
</tr>
<tr>
<td>Mesh3_Y</td>
<td>34,176</td>
<td>6,234</td>
<td>00:14:59</td>
<td>2</td>
</tr>
<tr>
<td>Mesh3_N</td>
<td>34,176</td>
<td>7,104</td>
<td>00:19:48</td>
<td>2</td>
</tr>
<tr>
<td>Mesh4_Y</td>
<td>136,704</td>
<td>5,526</td>
<td>01:18:56</td>
<td>2</td>
</tr>
<tr>
<td>Mesh4_N</td>
<td>136,704</td>
<td>9,206</td>
<td>02:05:03</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5_Y</td>
<td>546,816</td>
<td>14,791</td>
<td>17:19:00</td>
<td>2</td>
</tr>
<tr>
<td>Mesh5_N</td>
<td>546,816</td>
<td>14,898</td>
<td>18:03:36</td>
<td>2</td>
</tr>
</tbody>
</table>
A new command “SASW 1,” which implements the standard SA version, was applied. Overall, SC/T_Y (runs with the command) and SC/T_N (runs without the command) gave almost the same results for Cp and velocity profiles. From the results of the velocity profiles, the results of SC/Tetra agree well with the CFL3D (SA) results and are close to the experimental data. SC/Tetra also showed good grid convergence. SC/T_Y and SC/T_N have very close convergence cycles and running times. The only significant difference is that Mesh 4_N takes 3,700 cycles more than Mesh 4_Y. This is not sufficient to demonstrate that the command improves the convergence. Therefore, there is no strong evidence, based on this test case, to argue that adding the “SASW 1” command will improve the performance of the SA model.

For the SA turbulence model, the CFL3D turbulence setting instead 3 * kinematic viscosity was used for turbulent eddy viscosity at inlet. Again, both runs give the same results and are in excellent agreement with the CFL3D results.
5 Flow over NACA 0012 Airfoil

5.1 Problem Overview

In this chapter, we are analyzing the flow over a NACA0012 airfoil and focusing on the pressure coefficient. Structured and unstructured mesh is evaluated, the SST and SA turbulence models are tested, and the SC/Tetra results are compared with the CFL3D results.

5.2 Structured Meshes

A family of five pseudo-2d structured meshes is used in this investigation. They were obtained from the NASA Langley Turbulence Modeling Resource website. The meshes have a “C grid” structure; sizes are 113 x 33, 225 x 65, 449 x 129, 897 x 257, and 1793 x 513. These are referred to as meshes “Mesh S1” – “Mesh S5”. CFL3D data is available for verification from the same NASA website. In this case, three AoA (angles-of-attack) are considered: 0, 10, and 15 degrees. This is the sample mesh 449 X 129:

![Structured mesh grid 449 x 129. The left is the overall view of the mesh. The right is the close-up view of the airfoil.](image)

Figure 5-1: Structured mesh grid 449 x 129. The left is the overall view of the mesh. The right is the close-up view of the airfoil
The thickness of the first and the second prism layers for the structured meshes at different locations along the airfoil is shown in Table 5-1.

### Table 5-1: The thickness of the first and the second prism layer of structured mesh

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Leading, m</th>
<th>Half, m</th>
<th>Trailing, m</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>3.11496478e-5</td>
<td>8.61925030e-6</td>
<td>0.00983703</td>
</tr>
<tr>
<td></td>
<td>1.27703875e-4</td>
<td>1.57272072e-5</td>
<td>0.01504719</td>
</tr>
<tr>
<td>S2</td>
<td>8.02403792e-6</td>
<td>3.66417856e-6</td>
<td>0.00439786</td>
</tr>
<tr>
<td></td>
<td>2.31256099e-5</td>
<td>4.95073140e-6</td>
<td>0.00543916</td>
</tr>
<tr>
<td>S3</td>
<td>2.54595323e-6</td>
<td>1.69453672e-6</td>
<td>0.00208221</td>
</tr>
<tr>
<td></td>
<td>5.47808445e-6</td>
<td>1.96997668e-6</td>
<td>0.00231565</td>
</tr>
<tr>
<td>S4</td>
<td>9.50865285e-7</td>
<td>8.14297673e-7</td>
<td>0.00101351</td>
</tr>
<tr>
<td></td>
<td>1.59508823e-6</td>
<td>8.80086690e-7</td>
<td>0.00106871</td>
</tr>
<tr>
<td>S5</td>
<td>3.99989716e-7</td>
<td>4.02331352e-7</td>
<td>0.00049996</td>
</tr>
<tr>
<td></td>
<td>5.50875569e-7</td>
<td>4.13507223e-7</td>
<td>0.00051355</td>
</tr>
</tbody>
</table>

5.3 Analysis Conditions

The simulation of NACA0012 is a steady state simulation based on the Reynolds number 6e+6. The selected flow is incompressible flow with density changed to 1 kg/m3 and viscosity to 1.6667e-7 kg/(m.s). All the surfaces were registered as the Figure 5-2 for 0 degree AoA and Figure 5-3 for 10 and 15 degree AoA.

For 0 degree AoA, as the figure shows below, the inflow velocity is set at 1 m/s to match the Re number 6e+6, and turbulence properties: \( V_t \) is 5.0001e-7 m2/s only for SA, because the CFL3D SA runs used an inlet setting of turbulent eddy viscosity, which is 3*kinematic viscosity; this is repeated here, giving 5.0001e-7 m2/s.
However, for the 10 and 15 degree AoA, the same inlet region cannot be used as with the 0 degree AoA case. When the inflow is rotated to a non-zero AoA, this will lead to outflow across the top surface, so a free-slip boundary condition is no longer appropriate for the top surface. In order to fix the problem, previous top, bottom & outlet boundaries are added to the inlet, and a fixed pressure point is set near the airfoil, close to the location of maximum pressure. So, the inlet velocity is set to $1 \cos \theta$ m/s in the chord direction and $1 \sin \theta$ m/s in the perpendicular direction.
Up to 10000 cycles were applied for 0 degree AoA. However, more cycles were required for the 10 and 15 degree AoA; a final cycle of 60,000 was initially set for these cases. If 60,000 cycles were not enough, the solver was restarted with a final cycle of 100,000. Unfortunately, some simulations still did not reach the convergence criteria. In those cases, the solver was run until pressure forces were no longer changing. Some simulations diverged (see later in this chapter). These convergence problems may be caused by problems in the mesh quality, such as high aspect ratios. The convergence criterion is set to 1e-08, and under-relaxation is reduced to 0.5 for U, V, and W; 0.3 for P; matrix solver: P: max. Iterations = 1000; solver = AMGCG-STAB; convergence target = 1e-10; other variables: max. Iterations = 100; solver = MILUCG-STAB; convergence target = 1e-10. SST and SA turbulence models were used to test the difference.

The “CFL3D” results are from the NASA website. They are for the CFL3D code, using the SST or SA model and the 897X257 mesh (S4). All SST turbulence model runs used the command “SSTD LORE 0,” and all SA turbulence model runs used the command “SASW 1.” These changes to the default SC/Tetra SST and SA models are in agreement with the CFL3D runs. They also apply to the unstructured mesh runs later in this chapter. The run times do not scale uniformly with the number of cycles. This may be due to a wide variation in the number of iterations per cycle required for the matrix solver to converge.

5.4 SST Turbulence Model Results and Conclusions
(a) Mesh S1 and Mesh S2 0 degree AoA

(b) Mesh S3 0 degree AoA
(c) Mesh S1 and Mesh S2 10 degree AoA

(d) Mesh S3 10 degree AoA
Figure 5-4: Comparison of pressure coefficient on the airfoil surface of SST turbulence model

(e) Mesh S1 and Mesh S2 15 degree AoA

(f) Mesh S3 15 degree AoA
Flow over the NACA 0012 airfoil has been simulated using SC/Tetra and a family of five structured meshes downloaded from a NASA website. Converged solutions could only be obtained for the three coarsest meshes. However, the two finest meshes resulted in divergence, with huge values of pressure and velocity. Attempts to solve this issue, including various under-relaxation settings, and changing the location of the fixed pressure point for 10 and 15
degree AoA, were unsuccessful. These convergence issues may be caused by problems in the mesh quality with the finer meshes. For example, high aspect ratios close to the airfoil surface. From the results, the most accurate result was obtained using the middle mesh of the family of five, which was the finest mesh that reached convergence. And comparisons of peak (negative) Cp value show some indication of grid convergence, especially for the 0 degree angle-of-attack. The y+ values for the middle mesh range from 0.42 to 1.42. These are reasonable values for the use of the SST k-ω turbulence model. This may explain why the middle mesh gives the most accurate results. For the 0 degree AoA, there is good agreement between SC/Tetra (for the three meshes that converged) and the Cp CFL3D data. The range of differences of the peak value is from 5.08% to 0.72%, with both S2 and S3 just 0.72% less than the CFL3D data. This is a good indicator of grid convergence. For the additional tests, the range of differences of the peak value is from 4.11% to 0.24%. The S3 gives the best result. We would hope S4 would give the best result, but maybe the reason it does not is that its convergence status has big oscillations. For the 10 degree AoA, the range of differences is from 6.96% to 2.32%. The finest mesh to converge (S3) gives the best result. For the 15 degree AoA, the range of differences is from 13.94% to 2.47%. The S3 mesh again gives the best result. Therefore, as angle-of-attack increases, the difference between SC/Tetra results and the CFL3D data increases. The 10 and 15 degree AoA correspond to expected flow separation, where it is typically more difficult to arrive at an accurate CFD solution.

5.4.1 Additional Analysis for SST Turbulence Model

Additional tests were used for SST to address the convergence problem. The convergence criteria increased to 1e-6 (less strict than 1e-8) and the under-relaxation: 0.5 for
U, V, and W (Cradle Japan recommended no under-relaxation for P) were applied to test with meshes S1 – S4, with 0 degree AoA. It turns out that convergence was reached for the S2, S3, and S4 meshes. The S1 run still did not converge. If the convergence criteria is reduced to 1e-8, meshes S2 – S4 still get oscillations between 1e-6 and 1e-8. Note that these settings were only useful for 0 degree AoA. They did not help for 10 and 15 degree AoA.

(a) Mesh S2 0 degree AoA
Figure 5-5: Comparison of pressure coefficient of additional analysis results SST turbulence model
Table 5-4: Cycles to convergence and run times for additional analysis

<table>
<thead>
<tr>
<th>Mesh &amp; Angle of attack</th>
<th>Cycle</th>
<th>Running time, hr:min:sec</th>
<th>y+ value</th>
</tr>
</thead>
<tbody>
<tr>
<td>S2X0</td>
<td>4,631</td>
<td>00:14:57</td>
<td>0.94</td>
</tr>
<tr>
<td>S3X0</td>
<td>3,790</td>
<td>03:18:60</td>
<td>0.42</td>
</tr>
<tr>
<td>S4X0</td>
<td>7,543</td>
<td>28:27:58</td>
<td>0.20</td>
</tr>
</tbody>
</table>

Table 5-5: Comparisons of peak (negative) value of Cp additional test for additional analysis

<table>
<thead>
<tr>
<th></th>
<th>S2_Cp_0</th>
<th>S3_Cp_0</th>
<th>S4_Cp_0</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D</td>
<td>-0.413</td>
<td>-0.413</td>
<td>-0.413</td>
</tr>
<tr>
<td>Simulation</td>
<td>-0.404</td>
<td>-0.414</td>
<td>-0.430</td>
</tr>
<tr>
<td>Difference</td>
<td>-2.17%</td>
<td>+0.24%</td>
<td>+4.11%</td>
</tr>
</tbody>
</table>

In the additional tests, the simulations for meshes S2, S3, and S4 and 0 degree AoA now reach convergence. The most accurate results were obtained using the middle mesh. Furthermore, the graphs below show how the convergence progress for S3 was improved. Note: Mesh S4 gives a bigger oscillation for P and V than S3. This may be the reason why the results for S3 are more accurate. Changing the convergence criteria to 1e-06 does not affect the calculations, but it does allow the solver to terminate the run earlier.

Figure 5-6: Comparison of the convergence status between previous and additional analysis. The left indicates the Mesh 3 and the right indicates the previous Mesh 3
5.5 SA Turbulence Model Results and Conclusions

For SA turbulence model, analysis is only focusing on MeshS3

(a) Mesh S3 0 degree AoA
Figure 5-7: Comparison of pressure coefficient on the airfoil surface of SA turbulence model

(b) Mesh S3 10 degree AoA

(c) Mesh S3 15 degree AoA
Table 5-6: Cycles to convergence and run times for SA turbulence model

<table>
<thead>
<tr>
<th>Mesh &amp; Angle of attack</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, hr:min:sec</th>
<th>y+ value</th>
</tr>
</thead>
<tbody>
<tr>
<td>S3X0(SA)</td>
<td>57344</td>
<td>150,000</td>
<td>92:09:43</td>
<td>0.41</td>
</tr>
<tr>
<td>S3X10(SA)</td>
<td>57344</td>
<td>110,331</td>
<td>429:42:54</td>
<td>0.45</td>
</tr>
<tr>
<td>S3X15(SA)</td>
<td>57344</td>
<td>400,000</td>
<td>731:20:55</td>
<td>0.65</td>
</tr>
</tbody>
</table>

Note that S3x0 and S3x15 did not converge; results were obtained by averaging the oscillating values.

Table 5-7: Comparisons of peak (negative) value of Cp for SA turbulence model

<table>
<thead>
<tr>
<th></th>
<th>S1_Cp_0(SA)</th>
<th>S2_Cp_10(SA)</th>
<th>S3_Cp_15(SA)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D</td>
<td>-0.413</td>
<td>-5.66</td>
<td>-11.16</td>
</tr>
<tr>
<td>Simulation</td>
<td>-0.417</td>
<td>-5.47</td>
<td>-10.94</td>
</tr>
<tr>
<td>Difference</td>
<td>+0.96%</td>
<td>-3.35%</td>
<td>-1.97%</td>
</tr>
</tbody>
</table>

The SA turbulence model was used with mesh S3 only, comparing the results to the SA results and the CFL3D results using the SA model. There is excellent agreement between SC/Tetra SA results and CFL3D results for the 0 degree AoA. The difference in peak values is only 0.96%, though this is still slightly less accurate than the 0.7% difference achieved by SST. For 10 degree AoA, the difference in peak values using SA is 3.35%, whereas the SST peak was within 2.32%. For 15 degree AoA, SA gives a better result than SST, however. SA’s peak value is 1.97% away from the CFL3D peak, while SST’s peak was off by 2.47%. Thus, comparing the results of SST and SA, the SST model does slightly better overall, giving the more accurate peak (relative to CFL3D) for both 0 and 10 degree angle-of-attack.

5.6 Unstructured Meshes

In this part, six pseudo-2D unstructured meshes were developed for NACA 0012. The [Mesh generation by sweep] setting is used to get one layer of elements in the 3rd dimension.
Also, separate meshes were created for the outer and inner regions of the domain. Depending on the mesh, the size of the smallest octant varies from 4 mm to 0.125 mm. Element size is controlled by local octant size. The mesh name indicates the smallest octant. For example, “US2” is the unstructured mesh with octants down to 2 mm. The inner region and outer region are shown below. The geometry was built using the Solid Works CAD software with the inner domain radius of 1.5 m and overall dimensions of 50 m x 50 m. All surfaces were registered as the Figures 5-8 and 5-9 shown below:

Figure 5-8: The closed volume of center part

Figure 5-9: The closed volume of outer part
5.6.1 Outer Mesh

The same octant pattern was used for the outer region in both the coarse and fine meshes. Octants range from 1.024 m to 0.064 m.

![Figure 5-10: The octant levels of outer part](image)

Similar meshes are used for the outer region in both coarse and fine mesh cases; there are small differences due to a thinner 3rd dimension in fine mesh cases, which reduces the number of inner region octants. When the 3rd dimension is larger than the smallest octant, the octree will have multiple layers in the 3rd dimension. This can cause RAM requirements to be unmanageable during mesh generation. Also, outer meshes use ~ 13k elements and 13k nodes.

5.6.2 Inner Mesh

As an example, the inner region octants for the US4 mesh are shown below. The octant sizes for US4 range from 64 mm to 4 mm. Successive refinement was used, so the octree for US2, for example, was obtained by further refinement of a sub-region of the 4 mm octants in US4.
Table 5-8: Prism layer parameters, growth ratios are fixed at 1.1. (Note: The motivation for the “US0.125 20P” mesh was to reach a Y+ value as low as the finest structured mesh that converged (449x129))

<table>
<thead>
<tr>
<th>Mesh</th>
<th>1st prism layer</th>
<th>No. of layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>US4</td>
<td>0.5 mm</td>
<td>6</td>
</tr>
<tr>
<td>US2</td>
<td>0.4 mm</td>
<td>8</td>
</tr>
<tr>
<td>US1</td>
<td>0.2 mm</td>
<td>10</td>
</tr>
<tr>
<td>US0.5</td>
<td>0.1 mm</td>
<td>10</td>
</tr>
<tr>
<td>US0.25</td>
<td>0.05 mm</td>
<td>10</td>
</tr>
<tr>
<td>US0.125</td>
<td>0.02 mm</td>
<td>10</td>
</tr>
<tr>
<td>US0.125 20P</td>
<td>0.002mm</td>
<td>20</td>
</tr>
</tbody>
</table>

The US4 mesh inner part has 59k elements, 62k nodes, and 1 layer of elements in the 3rd dimension (Z)

Figure 5-11: Close up of MeshUS4X0 AoA (Note that 6 prism layers on airfoil surface; 1st prism = 0.5mm; growth ratio = 1.1)

5.6.3 Merged Mesh

The US4 mesh has a total of 190k elements, 190k nodes, and 1 layer of elements in the 3rd dimension (Z)
5.7 Analysis Conditions

For the unstructured mesh, it is also a steady state simulation with maximum cycle of 400,000. The convergence criterion is $1e$-8. If calculated variables are oscillating, results are averaged over an appropriate number of cycles. Under-relaxation settings are the same as the structured mesh. The inlet velocity is 1 m/s perpendicular to inlet face, and the $v_t$ is $5.0001e$-7 for turbulence properties only for the SA turbulence model (the same as with the structured meshes). The outlet condition is static pressure. Also, the airfoil is set to no-slip boundary condition. The up_wall and low_wall were free-slip. SST and SA turbulence models were used, and the SST turbulence model turns off the Wilcox correction by manually adding “SSTD LORE 0/” to the S-file. The “Discontinuous mesh interface” setting was used to link the outer and inner mesh regions, applied to outer_side and inner_side surfaces.
5.8 SST Turbulence Model Results and Conclusions

(a) Mesh US4–Mesh US0.5 0 degree AoA
(b) Mesh US0.25–Mesh US0.125_20p 0 degree AoA

(c) Mesh US4–Mesh US0.5 10 degree AoA
(d) Mesh US0.25~Mesh US0.125_20p 10 degree AoA

(e) Mesh US4~Mesh US0.5 15 degree AoA
Figure 5-13: Comparison of pressure coefficient on the airfoil surface SST turbulence model Mesh US0.25 was tested to see the difference between the SST and SA models.
Figure 5-14: Comparison of pressure coefficient of results SST and SA turbulence models

(b) Mesh US0.25 10 degree AoA

(c) Mesh US0.25 15 degree AoA
Table 5-9: Cycles to convergence and run times for unstructured mesh. (Note that * The unusually long run times for US2 and US1 may be due to underperforming compute nodes on the cluster machine)

<table>
<thead>
<tr>
<th>Mesh Size and angle-of-attack</th>
<th>Number of mesh elements</th>
<th>Number of cycles</th>
<th>Running time (hr:min:sec)</th>
<th>Y+ value</th>
</tr>
</thead>
<tbody>
<tr>
<td>US4X0</td>
<td>189,793</td>
<td>18,890</td>
<td>06:02:06</td>
<td>121.41</td>
</tr>
<tr>
<td>US4X10</td>
<td>66,026</td>
<td>31,201</td>
<td>23:42:34</td>
<td>104.33</td>
</tr>
<tr>
<td>US4X15</td>
<td>400,000</td>
<td>147,366</td>
<td>119:16:55</td>
<td>89.78</td>
</tr>
<tr>
<td>US2X0</td>
<td>222,640</td>
<td>45,703</td>
<td>88:19:02*</td>
<td>93.17</td>
</tr>
<tr>
<td>US2X10</td>
<td>32,674</td>
<td>51,201</td>
<td>94:26:56*</td>
<td>89.20</td>
</tr>
<tr>
<td>US2X15</td>
<td>147,366</td>
<td>135,178</td>
<td>168:30:06*</td>
<td>75.06</td>
</tr>
<tr>
<td>US1X0</td>
<td>287,178</td>
<td>80,060</td>
<td>54:09:59</td>
<td>48.38</td>
</tr>
<tr>
<td>US1X10</td>
<td>383,554</td>
<td>87,856</td>
<td>63:48:05</td>
<td>45.84</td>
</tr>
<tr>
<td>US1X15</td>
<td>117,714</td>
<td>107,16:06*</td>
<td>80:37:28</td>
<td>41.71</td>
</tr>
<tr>
<td>US0.5X0</td>
<td>748,598</td>
<td>64,377</td>
<td>87:37:07</td>
<td>11.53</td>
</tr>
<tr>
<td>US0.5X10</td>
<td>83,191</td>
<td>83,191</td>
<td>108:12:51</td>
<td>10.97</td>
</tr>
<tr>
<td>US0.5X15</td>
<td>141,937</td>
<td>141,937</td>
<td>185:25:24</td>
<td>10.11</td>
</tr>
<tr>
<td>US0.125X0</td>
<td>985,318</td>
<td>72,949</td>
<td>143:17:49</td>
<td>4.59</td>
</tr>
<tr>
<td>US0.125X10</td>
<td>60,467</td>
<td>60,467</td>
<td>116:07:10</td>
<td>4.38</td>
</tr>
<tr>
<td>US0.125X15</td>
<td>257,595</td>
<td>257,595</td>
<td>538:45:45</td>
<td>4.01</td>
</tr>
<tr>
<td>US0.125X0(20P)</td>
<td>1,178,411</td>
<td>31,320</td>
<td>92:46:32</td>
<td>0.44</td>
</tr>
<tr>
<td>US0.125X10(20P)</td>
<td>42,665</td>
<td>42,665</td>
<td>129:03:46</td>
<td>0.42</td>
</tr>
<tr>
<td>US0.125X15(20P)</td>
<td>88,099</td>
<td>88,099</td>
<td>280:35:41</td>
<td>0.38</td>
</tr>
<tr>
<td>(SA)US0.25X0</td>
<td>748,598</td>
<td>25,151</td>
<td>34:03:22</td>
<td>11.55</td>
</tr>
<tr>
<td>(SA)US0.25X10</td>
<td>58,975</td>
<td>58,975</td>
<td>73:45:29</td>
<td>11.07</td>
</tr>
<tr>
<td>(SA)US0.25X15</td>
<td>88,467</td>
<td>88,467</td>
<td>107:11:29</td>
<td>10.13</td>
</tr>
</tbody>
</table>

However, The US4X15 case did not converge; results were obtained by averaging the oscillating values. For example, here are convergence status and pressure graphs for US4 x 15:
Figure 5-15: The convergence status of MeshUS4X15 AoA. The left indicates the convergence status and the right indicates the summation of pressure on airfoil.

Table 5-10: Comparisons of peak (negative) value of Cp for unstructured mesh

<table>
<thead>
<tr>
<th></th>
<th>0 degree AoA</th>
<th>10 degree AoA</th>
<th>15 degree AoA</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D data</td>
<td>-0.413</td>
<td>-0.413</td>
<td>-0.413</td>
</tr>
<tr>
<td>Simulation</td>
<td>-0.409</td>
<td>-0.410</td>
<td>-0.415</td>
</tr>
<tr>
<td>Difference</td>
<td>-0.96%</td>
<td>-0.72%</td>
<td>+0.48%</td>
</tr>
</tbody>
</table>

87
There is excellent agreement between SC/Tetra results and the CFL3D data for the 0 degree AoA. There are seven different meshes that were tested, and the results are all very close to the CFL3D data. The differences for all meshes are on the order of 1%. The result for the US0.25 mesh actually agrees exactly with the data. For the 10 degree AoA, the range of differences for the seven meshes is from 10.89% to 1.07%. For SST, the worst result is for the coarsest mesh (US4; y+ = 104), and the best result of 1.07% is for the US1 mesh; y+ = 45.84. For SA, the best result is for the US0.125 mesh (y+ = 11.07). These results indicate both grid convergence and also improved performance as the y+ value approaches the recommended range for the SST $k$-$\omega$ turbulence model. Furthermore, the US0.125 mesh was also tested with 20 prism layers inserted, rather than the previous 10 layers. At the same time, the thickness of the 1st prism layer was reduced by an order of magnitude to 0.002 mm. The aim was to test a y+ value similar to the finest structured mesh that converged. The total thickness of prism layers with the 20 layers was 0.1145 mm; with the 10, thicker layers, it was 0.3817 mm. Overall, the accuracy was similar for these two US0.125 meshes. This may indicate that grid convergence had been reached already. The greater total thickness of prisms may also help the accuracy when the 10 layers are used. Moreover, for the 15 degree AoA, the range of differences is from 35.5% to 2.15%. The best result was given by the US0.25 (SA) mesh. The

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D data</td>
<td>-10.9</td>
<td>-10.9</td>
<td>-10.9</td>
<td>-10.9</td>
</tr>
<tr>
<td>Simulation</td>
<td>-7.03</td>
<td>-9.55</td>
<td>-10.42</td>
<td>-10.52</td>
</tr>
<tr>
<td>Difference</td>
<td>-35.50%</td>
<td>-12.38%</td>
<td>-4.40%</td>
<td>-3.48%</td>
</tr>
<tr>
<td></td>
<td>US0.25_Cp_15 (SST)</td>
<td>US0.25_Cp_0 (SA)</td>
<td>US0.125_Cp_15</td>
<td>US0.125_Cp_15_20P</td>
</tr>
<tr>
<td>CFL3D data</td>
<td>-10.9</td>
<td>-11.16</td>
<td>-10.9</td>
<td>-10.9</td>
</tr>
<tr>
<td>Simulation</td>
<td>-10.43</td>
<td>-10.92</td>
<td>-10.51</td>
<td>-10.47</td>
</tr>
<tr>
<td>Difference</td>
<td>-4.31%</td>
<td>-2.15%</td>
<td>-3.57%</td>
<td>-3.94%</td>
</tr>
</tbody>
</table>


US0.5 mesh (within 3.48% of CFL3D) gave the best result for the SST k-ω model. Therefore, results for the SST k-ω and SA turbulence models were directly compared using the US0.25 mesh. The SA results were found to be more accurate for 10 and 15 degree AoA. However, SST k-ω gives the best result for 0 degree AoA. The lower y+ meshes for SST give a better result for 10 degree AoA than SA with US0.25. However, for 15 degree AoA, SA with US0.25 gives the best result of all. The y+ values in these simulations were on the order of 10, which is too high to expect the best performance from these “low Reynolds number” turbulence models. Both SST and SA have been widely verified for flow over an airfoil (refer to the previously mentioned NASA website, for example), but the generally recommended range of application is $y^+ < 3$ for SST and $y^+ < 5$ for SA.

5.9 VB Application

SC/Tetra operations can be automatically controlled using the Visual Basic programming language. An automated Visual Basic Application was developed to run the many unstructured mesh cases included in this report. With 6 meshes, 3 angles-of-attack, and 2 different turbulence models, automated mesh generation and simulation execution saved a great deal of time. The graphic on the following page illustrates the simulation control procedure using the Visual Basic application. First of all, The VBA method (using an Excel application) can be used to automatically rotate the inner region mesh to achieve different angles-of-attack. Merging of inner and outer meshes, as well as launching of the simulation, can also be done automatically. Furthermore, the computer will set up and solve all the cases, one-by-one. Last but not least, when each case is finished, the VBA application automatically outputs selected values from the L-file into the Excel worksheet.
5.10 Conclusions of Structured and Unstructured Meshes

It is difficult to directly compare the structured and unstructured mesh results for the SST model since the ranges of $y+$ values are quite different. The converged structure runs cover a $y+$ range of around 0.5 – 4, whereas the unstructured runs cover around 4.5 – 100. The S1 mesh ($y+ \sim 4$) and US0.125 mesh ($y+ \sim 4.5$) offer the best opportunity for a fair comparison: First, the unstructured mesh results are significantly more accurate for all 3 AoA, with $C_p$ peak differences (versus the NASA results for CFL3D) of 1.2 – 3.6%. The structured mesh peak differences are 5.1 – 13.9%. Second, the unstructured mesh run times are, however, around one order of magnitude longer than the structured mesh run times. This is caused by the much larger number of elements; 986k versus 4k. Third, the structured mesh elements have higher aspect ratios than the unstructured elements. This permits the low number of elements in the structured mesh, but may also contribute to the higher errors. This can be considered a trade-off of sorts. Mesh S3, a structured mesh with a much lower $y+$ than US0.125, can be run in a similar amount of time and gives comparable accuracy. However, the structured meshes have much more serious convergence problems than the unstructured meshes. This may be another consequence of the high aspect ratios of the elements, particularly near the surface of the airfoil. Convergence tends to become more difficult for lower $y+$ values, however, so this should also be taken into account. Comparing the complete set of structured and unstructured results, irrespective of $y+$ values, the best results are given by unstructured meshes for 0 and 10 degree AoA (0% for US0.25 and 1.07% for US1, respectively). A structured mesh gives the best results for 15 degree AoA, however (2.47% for S3).

The SA turbulence model was tested with just one structured mesh and one unstructured mesh: S3; $y+ \sim 0.5$; no. of elements = 57k; US0.25; $y+ \sim 11$; no. of elements = 749k. Due to
the wide difference in y+ values, we cannot draw any firm conclusions by comparing these results. Furthermore, the y+ for US0.25 is above the generally recommended range of y+ < 5. With these factors in mind, we will note that the best SA results for the Cp peak are given by: Structured mesh S3 for 0 and 15 degree AoA (0.96% and 1.97% differences, respectively); Unstructured mesh US0.25 for 10 degree AoA (1.94% differences). The lower y+ value is likely to help the accuracy of the structured results, but conversely, as with SST, the higher aspect ratios of the structured mesh elements will tend to reduce accuracy and slow down convergence.
6 Flow over NACA 4412 Airfoil

6.1 Problem Overview

The simulation of flow over NACA4412 is the focus of this chapter. The goal of this chapter is to predict the flow separation near the trailing edge. The effects of octant sizes and turbulence model including the SST and SA turbulence models are concentrated on unstructured mesh. The pressure coefficient and velocity profile results of SC/Tetra are compared with CFL3D and experimental results for both structured and unstructured meshes.

6.2 Structured Meshes

A family of five pseudo-2d structured meshes is used in this investigation. All meshes were obtained from the NASA Langley Turbulence Modeling website. All the structured meshes have a “C grid” structure; sizes are 113 x 33, 225 x 65, 449 x 129, 897 x 257, and 1793 x 513. These will be referred to as meshes “Mesh S1” – “Mesh S5.” Experimental data is available for verification from the same NASA website. We only focus on the 13.87 degrees AoA for this case. All the surface definitions are shown in the Figure 6-2; all the surface around the airfoil is registered as inlet. See sample mesh 449 x 129 grid as in Figure 6-1 shown below:
6.3 Analysis Conditions

In this case, it is also a steady state simulation dependent on the Reynolds number $1.52e+6$ with a maximum of 20,000 cycles, except 60,000 for Mesh 5 because it uses the finest mesh. The airfoil face is set to smooth no-slip wall, and the inlet velocity is set to $29.8825 \text{ m/s}$ in the X direction and $7.3785 \text{ m/s}$ in the Y direction, with turbulence properties 0.0001 for $k$ and $e$. In order to match the Mach number (0.09) and Re number, the coming flow is
incompressible air with density changed to 1.0 kg/m³ and viscosity to 2.025x10^-5 Pa-s. Because all the surfaces are registered to inlet, so a fixed static pressure point is applied at -0.246, -0.018, 0 near the airfoil. The convergence criterion is 1e-6. Also, SST k-ω and SA turbulence models are applied with the command “SSTD LORE 0/” and “SASW 1,” respectively.

6.4 SST Turbulence Model Results and Conclusions

When analyzing CFD results, we must divide the u/u_ref velocities by 0.93; this is needed to apply normalization in terms of an experimental reference location below and behind the airfoil. And all velocity profile lines are located in the field surrounding the trailing edge region of the airfoil: x/c (Chord length) = 0.675, 0.731, 0.786, 0.842, 0.897, and 0.953.

The compared results are Exp_Cp (experimental data), and CFL3D_Cp (NASA CFL3D code), using the SST model and equivalent of “LORE 0,” with the second finest mesh (897 x 257).
(a) Mesh S1

(b) Mesh S2
(c) Mesh S3

(d) Mesh S4
Figure 6-3: Comparison of pressure coefficient on the airfoil surface of SST turbulence model

(a) Mesh S1

(e) Mesh S5
Figure 6-4: Comparison of velocity profile on different locations of SST turbulence model
From the results, the most accurate Cp results were obtained using Mesh S5, which was the finest mesh. The Cp peak for Mesh S5 is only 0.12% away from the CFL3D(SST) peak. The y+ value for Mesh S5 is 0.12. It is easily small enough for the use of the SST turbulence model. And comparison of peak (negative) Cp value shows some indication of grid convergence (recommended range is y+ < 3). Moreover, the velocity profiles also indicate grid convergence. Overall, the velocity profiles are in good agreement with CFL3D. The best match is reached with Mesh S3, but Mesh S4 and Mesh S5 also do well. Furthermore, to remedy the convergence problems for Mesh S1, S2, and S5, we tried several methods: Non-default under-relaxation settings; 0.5 for U, V, W; Non-default pseudo-timestep relaxation settings: 5 for U, V, W; Modified matrix solver settings: P: max. Iterations = 1000; solver = AMGCG-STAB; convergence target = 1e-10. Other variables: max. Iterations = 100; solver = MILUCG-STAB; convergence target = 1e-10 (Note: These matrix solver settings result in much longer solution times, so they are not recommended.) However, none of these methods helped with the convergence problems.

6.5 SA Turbulence Model Results and Conclusions

The SA turbulence model was applied to Mesh S3 only. Note: CFL3D_Cp = NASA CFL3D code, using standard SA model, (equivalent of “SASW 1”), with second finest mesh (897x257).
Figure 6-5: Comparison of pressure coefficient on the airfoil surface of SA turbulence model

Figure 6-6: Comparison of velocity profile on the different locations of SA turbulence model
The convergence status of Meshes S1, S2, and S5 were oscillating and did not reach the convergence criteria. Results were obtained by averaging the oscillating values.

Table 6-1: Cycles to convergence and run time for SST and SA turbulence model

<table>
<thead>
<tr>
<th>Mesh &amp; Angle of attack</th>
<th>Elements</th>
<th>Cycles</th>
<th>Running time (h:m:s)</th>
<th>y+ value</th>
<th>Run Platform</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshS1(SST)</td>
<td>3,581</td>
<td>20,000</td>
<td>00:23:59</td>
<td>2.18</td>
<td>1</td>
</tr>
<tr>
<td>MeshS2(SST)</td>
<td>14,336</td>
<td>20,000</td>
<td>01:09:15</td>
<td>0.97</td>
<td>2</td>
</tr>
<tr>
<td>MeshS3(SST)</td>
<td>57,344</td>
<td>11,198</td>
<td>02:04:54</td>
<td>0.48</td>
<td>2</td>
</tr>
<tr>
<td>MeshS3(SA)</td>
<td>57,344</td>
<td>15,193</td>
<td>02:33:29</td>
<td>0.48</td>
<td>2</td>
</tr>
<tr>
<td>MeshS4(SST)</td>
<td>229,376</td>
<td>14,021</td>
<td>06:56:37</td>
<td>0.24</td>
<td>2</td>
</tr>
<tr>
<td>MeshS5(SST)</td>
<td>917,504</td>
<td>60,000</td>
<td>115:49:35</td>
<td>0.12</td>
<td>2</td>
</tr>
</tbody>
</table>

Table 6-2: Comparisons of the peak (negative) value of Cp with CFL3D results

<table>
<thead>
<tr>
<th>Mesh &amp; Angle of attack</th>
<th>S1</th>
<th>S2</th>
<th>S3</th>
<th>S3(SA)</th>
<th>S4</th>
<th>S5</th>
</tr>
</thead>
<tbody>
<tr>
<td>SC/T</td>
<td>-4.2780</td>
<td>-6.1044</td>
<td>-6.7991</td>
<td>-7.3716</td>
<td>-6.8281</td>
<td>-6.9029</td>
</tr>
<tr>
<td>Difference</td>
<td>-37.95%</td>
<td>-11.46%</td>
<td>-1.38%</td>
<td>-0.23%</td>
<td>-0.96%</td>
<td>0.12%</td>
</tr>
</tbody>
</table>

Figure 6-7: Convergence status sample: Mesh S1
The SA turbulence model was tested only with Mesh S3. The results were compared to the CFL3D SA results. From the results, the Cp peak is within 0.23% of the CFL3D peak, which is a closer match than achieved by the SST run for Mesh S3. Both y+ values were 0.48. The recommended range for SA is y+ < 5. Likewise, the velocity profile agrees well the CFL3D SA profile. Both SC/T and CFL3D profiles are different from the experimental data. In terms of model validation, the key goal is demonstrating good agreement of SC/T with CFL3D.

6.6 Unstructured Meshes

In this case, there is only experimental data for a single angle-of-attack. There was, therefore, no need to divide the domain into inner and outer regions as in Chapter 3 for NACA0012. The mesh was generated for the entire domain at once. All unstructured mesh settings followed Tomohiro Irie’s paper (Irie (2011)). So, meshes are pseudo-2D, with one element in the Z-direction. The domain dimension is a 10 m diameter cylinder (Note: We used a domain size as small as possible, without significantly affecting the results. A larger domain size would have resulted in more elements, so that mesh generation was demanding and run times were longer.) The octant sizes, number of prism layers, and prism growth ratio are changed to examine the effects of these changes on the results.
Different octant sizes were used to establish grid convergence.

<table>
<thead>
<tr>
<th>Mesh name</th>
<th>Minimum octant size (mm)</th>
<th>Downstream direction-range of the minimum octant size (No. of octants)</th>
<th>No. of nodes</th>
<th>No. of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshUS4</td>
<td>4</td>
<td>40</td>
<td>444,936</td>
<td>441,928</td>
</tr>
<tr>
<td>MeshUS2</td>
<td>2</td>
<td>40</td>
<td>577,826</td>
<td>570,523</td>
</tr>
<tr>
<td>MeshUS11</td>
<td>1</td>
<td>40</td>
<td>802,200</td>
<td>780,261</td>
</tr>
<tr>
<td>MeshUS12</td>
<td>1</td>
<td>80</td>
<td>839,820</td>
<td>817,881</td>
</tr>
<tr>
<td>MeshUS13</td>
<td>1</td>
<td>160</td>
<td>991,186</td>
<td>969,247</td>
</tr>
<tr>
<td>MeshUS0.5</td>
<td>0.5</td>
<td>40</td>
<td>1,153,478</td>
<td>1,090,774</td>
</tr>
</tbody>
</table>

The "base mesh" used octants down to 8 mm, and then successive refinements were applied around the airfoil to get four more meshes. The meshes used up to 15 prism layers on the airfoil surface; increase ratio to 1.1 mm. See sample Mesh US11:
Table 6-4: Unstructured mesh sizes and prism information

<table>
<thead>
<tr>
<th>Mesh</th>
<th>1st prism layer (mm)</th>
<th>No. of layers</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshUS4</td>
<td>0.8</td>
<td>4</td>
</tr>
<tr>
<td>MeshUS2</td>
<td>0.4</td>
<td>6</td>
</tr>
<tr>
<td>MeshUS11</td>
<td>0.2</td>
<td>10</td>
</tr>
<tr>
<td>MeshUS12</td>
<td>0.2</td>
<td>10</td>
</tr>
<tr>
<td>MeshUS13</td>
<td>0.2</td>
<td>10</td>
</tr>
<tr>
<td>MeshUS0.5</td>
<td>0.1</td>
<td>15</td>
</tr>
</tbody>
</table>

Figure 6-10: Different meshes around the leading edge: (a) Mesh US4, (b) Mesh US2, (c) Mesh US11, (d) Mesh US0.5
Figure 6-11: Different meshes around the airfoil section: (a) Mesh US11, (b) Mesh US12, (c) Mesh US13
6.7 Analysis Conditions

The analysis condition for unstructured mesh is the same as the analysis condition for the structured mesh. For the additional analysis Section 3, we are testing a different turbulence model: Standard k-ε (“SKE”), linear low-Reynolds-number k-EPS model (“AKN”), Spalart-Allmaras (“SA”). We used the same sets of turbulence models as the primary references for the two sections of this report (NASA site for structured mesh section and Irie's report for the unstructured mesh section (Irie (2011))).

6.8 SST Turbulence Model Results and Conclusions

(a) Mesh US4
(b) Mesh US2

(c) Mesh US11
(d) Mesh US12

(e) Mesh US13
Figure 6-12: Comparison of pressure coefficient on the airfoil surface of SST turbulence model
(b) Mesh US2

(c) Mesh US11
Figure 6-13: Comparison of velocity profiles on different locations of SST turbulence model

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, h:m:s</th>
<th>Run Platform</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshUS4</td>
<td>441,928</td>
<td>30,000</td>
<td>10:36;34</td>
<td>2</td>
<td>41.35</td>
</tr>
<tr>
<td>MeshUS2</td>
<td>570,523</td>
<td>30,000</td>
<td>23:55:37</td>
<td>2</td>
<td>22.73</td>
</tr>
<tr>
<td>MeshUS11</td>
<td>780,261</td>
<td>4,589</td>
<td>02:44:40</td>
<td>2</td>
<td>11.40</td>
</tr>
<tr>
<td>MeshUS12</td>
<td>817,881</td>
<td>4,917</td>
<td>04:20:34</td>
<td>2</td>
<td>11.38</td>
</tr>
<tr>
<td>MeshUS13</td>
<td>969,247</td>
<td>5,396</td>
<td>05:42:38</td>
<td>2</td>
<td>11.38</td>
</tr>
<tr>
<td>MeshUS0.5</td>
<td>1,090,774</td>
<td>6,394</td>
<td>07:49:14</td>
<td>2</td>
<td>5.81</td>
</tr>
</tbody>
</table>

Table 6-6: Comparisons of the peak (negative) value of Cp with CFL3D results for SST turbulence model

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Difference</td>
<td>-36.75%</td>
<td>-3.16%</td>
<td>+4.42%</td>
<td>+4.07%</td>
<td>+4.10%</td>
<td>+5.65%</td>
</tr>
</tbody>
</table>
The peak Cp results show agreement with CFL3D (SST) when the minimum octant size is 1 mm or less. The peak values for Mesh US11, 12, and 13 are all within 5% of the CFL3D peak. Even though the Mesh US2 is 3.06% of the CFL3D peak, the pressure on the top surface is different. Moreover, the mesh y+ values range from 41.35 to 5.81. The worst Cp result is given by the coarsest mesh, Mesh US4, because of its large y+ value. The best result is for Mesh US12 (y+ =11.38), where the peak is within 4.07% of the CFL3D value. The finest mesh gives an error of 5.65%. Thus, there is some evidence of grid convergence and the meshes with y+ around 11 all give a better Cp peak. Furthermore, the velocity profiles also indicate grid convergence. Overall, the velocity profiles are in good agreement with CFL3D. The best match is reached with Mesh US11, 12, and 13.

6.8.1 Additional Analysis [1]: Further Mesh Refinement and Conclusions

In the additional analysis, the SST k-ω turbulence model is used, and the minimum octant size is 1 mm. Prism layer elements are fixed to 10 prism layers; growth ratio is 1.1; first prism layer is 0.2 mm. The coverage range of finer octants around the airfoil surface is reduced.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Range of the finest octant (No. of octants)</th>
<th>No. of nodes</th>
<th>No.of elemnts</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>+X</td>
<td>-X</td>
<td>+Y</td>
</tr>
<tr>
<td>MeshUS14</td>
<td>20</td>
<td>5</td>
<td>10</td>
</tr>
<tr>
<td>MeshUS15</td>
<td>10</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>MeshUS16</td>
<td>4</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>MeshUS17</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

(a) (b)
Figure 6-14: Close-up views of different ranges of octant size for computational meshes in the airfoil region: (a) 1 mm x 20, (b) 1 mm x 10, (c) 1 mm x 4, (d) 1 mm x 2
(a) Mesh US14

(b) Mesh US15
(c) Mesh US16

(d) Mesh US17
Figure 6-15: Comparison of pressure coefficient on the airfoil surface of additional analysis [1]

(a) Mesh US14

(b) Mesh US15
Figure 6-16: Comparison of velocity profiles on different locations of additional analysis [1]
Table 6-8: Cycles to convergence and run times for additional analysis [1]

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, h:m:s</th>
<th>Run Platform</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshUS14</td>
<td>530,663</td>
<td>5,516</td>
<td>04:05:17</td>
<td>2</td>
<td>11.36</td>
</tr>
<tr>
<td>MeshUS15</td>
<td>463,515</td>
<td>6,472</td>
<td>02:20:07</td>
<td>2</td>
<td>11.29</td>
</tr>
<tr>
<td>MeshUS16</td>
<td>441,137</td>
<td>18,276</td>
<td>05:59:22</td>
<td>2</td>
<td>11.23</td>
</tr>
<tr>
<td>MeshUS17</td>
<td>435,131</td>
<td>30,000</td>
<td>16:42:12</td>
<td>2</td>
<td>11.01</td>
</tr>
</tbody>
</table>

Table 6-9: Comparisons of the peak (negative) value of Cp with CFL3D results for additional analysis [1]

<table>
<thead>
<tr>
<th></th>
<th>US14</th>
<th>US15</th>
<th>US16</th>
<th>US17</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D</td>
<td>-6.8947</td>
<td>-6.8947</td>
<td>-6.8947</td>
<td>-6.8947</td>
</tr>
<tr>
<td>SC/T</td>
<td>-7.1456</td>
<td>-6.8372</td>
<td>-6.4365</td>
<td>-6.1649</td>
</tr>
<tr>
<td>Difference</td>
<td>+3.63%</td>
<td>-0.83%</td>
<td>-6.64%</td>
<td>-10.58%</td>
</tr>
</tbody>
</table>

In additional analysis [1], reduction of the coverage range of the finest octants was tested. The Cp results become poor when the downstream range is reduced to 4. Also, the velocity results become poor when the downstream range is reduced to 20. (Note: In Irie’s paper, additional analysis [1] indicated that the range of the finest octant size could be reduced to 4 without losing accuracy.) The results vary a lot despite meshes US14 to US17 having similar y+ values. The best Cp result comes from Mesh US15, which has a coverage range of 10. Its Cp peak is within 0.83% of the CFL3D value. The worst result is from Mesh US17, which has a coverage range of just 2. Its Cp peak is 10.58% away from the CFL3D peak. Therefore, even though the octant sizes and prism layers are the same for these 4 meshes, it is clear that other factors can be important. In particular, it is vital to have a sufficiently wide coverage region around the airfoil for the finest octants.
6.8.2 Additional Analysis [2]: Varied Prism Layer Settings and Conclusions

In accordance with additional analysis [1], the downstream range of the finest octant size is kept at 20. And the number of prism layers and the growth ratio are now varied. The thickness of the first prism layer remains the same (0.1mm).

Table 6-10: Parameters of prism layer elements

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Growth ratio</th>
<th>No. of layers</th>
<th>Total thickness (mm)</th>
<th>No. of nodes</th>
<th>No. of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshUSR1</td>
<td>1.1</td>
<td>15</td>
<td>3,177</td>
<td>572,048</td>
<td>540,399</td>
</tr>
<tr>
<td>MeshUSR2</td>
<td>1.1</td>
<td>10</td>
<td>1,593</td>
<td>557,600</td>
<td>535,661</td>
</tr>
<tr>
<td>MeshUSR3</td>
<td>1.3</td>
<td>9</td>
<td>3,201</td>
<td>548,598</td>
<td>528,601</td>
</tr>
<tr>
<td>MeshUSR4</td>
<td>1.5</td>
<td>7</td>
<td>3,217</td>
<td>540,620</td>
<td>524,507</td>
</tr>
<tr>
<td>MeshUSR5</td>
<td>2</td>
<td>5</td>
<td>3,100</td>
<td>533,236</td>
<td>521,007</td>
</tr>
</tbody>
</table>

(a)          (b) 

(c)          (d)
Figure 6-17: Computational mesh of different parameters of prism elements: (a) Mesh USR1, (b) Mesh USR2, (c) Mesh USR3, (d) Mesh USR4, (e) Mesh USR5

(a) Mesh USR1
Figure 6-18: Comparison of pressure coefficient on the airfoil surface of additional analysis [2]
(a) Mesh USR1

(b) Mesh USR2
Figure 6-19: Comparison of velocity profile on different locations of additional analysis [2]

Table 6-11: Cycles to convergence and run times for additional analysis [2]

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, h:m:s</th>
<th>Run Platform</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td>MeshUSR1</td>
<td>540,399</td>
<td>5,398</td>
<td>04:08:45</td>
<td>2</td>
<td>5.69</td>
</tr>
<tr>
<td>MeshUSR2</td>
<td>535,661</td>
<td>6,764</td>
<td>05:01:11</td>
<td>2</td>
<td>5.57</td>
</tr>
<tr>
<td>MeshUSR3</td>
<td>528,601</td>
<td>4,798</td>
<td>02:02:52</td>
<td>2</td>
<td>5.67</td>
</tr>
<tr>
<td>MeshUSR4</td>
<td>524,507</td>
<td>4,504</td>
<td>03:48:46</td>
<td>2</td>
<td>5.64</td>
</tr>
<tr>
<td>MeshUSR5</td>
<td>521,007</td>
<td>3,983</td>
<td>01:36:23</td>
<td>2</td>
<td>5.57</td>
</tr>
</tbody>
</table>

Table 6-12: Comparisons of the peak (negative) value of Cp with CFL3D results for additional analysis [2]

<table>
<thead>
<tr>
<th></th>
<th>USR1</th>
<th>USR2</th>
<th>USR3</th>
<th>USR4</th>
<th>USR5</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D</td>
<td>-6.8947</td>
<td>-6.8947</td>
<td>-6.8947</td>
<td>-6.8947</td>
<td>-6.8947</td>
</tr>
<tr>
<td>SC/T</td>
<td>-6.9921</td>
<td>-6.8420</td>
<td>+7.0769</td>
<td>+7.0922</td>
<td>+7.1642</td>
</tr>
<tr>
<td>Difference</td>
<td>-1.14%</td>
<td>-0.76%</td>
<td>+2.64%</td>
<td>+2.86%</td>
<td>+3.90%</td>
</tr>
</tbody>
</table>

In additional analysis [2], we investigate the effects of varying the number of prism layers and the growth rate. The thickness of the first prism layer is kept at 0.1 mm. The
downstream coverage of the finest octants is fixed at 20 octants, considering the Cp and velocity results from additional analysis [1]. Analysis of the velocity results shows that good accuracy is given by the meshes with total prism thicknesses greater than 3 mm. However, analysis also shows that the rest of the Cp profiles are worst for Mesh USR2; only the peak is better, within 0.76%. We thus conclude that the thickness of the first prism and the total thickness of all the prism layers are more important factors for accuracy than the number of prism layers and the growth ratio.

6.8.3 Additional Analysis [3]: Different Turbulence Models and Conclusions

Different turbulence models were tested using Mesh US11 to determine which model(s) gave the most accurate results for flow over the airfoil. Mesh US11 was selected as the basis for comparison as it combines a good Cp result with good velocity profiles and a relatively a low number of elements. The additional turbulence models are: (as used in reference [3]) Standard k-ε; AKN k-ε (“AKN”); Spalart-Allmaras (“SA”). In this part, streamlines are now provided to investigate which turbulence models are able to resolve the expected flow separation near the trailing edge. Pressure coefficient and velocity profiles of the SST, SKE and AKN models are also compared with experimental data and CFL3D SST results. SA profiles are compared with experimental data and CFL3D SA results.

(a) SST
(b) AKN
Figure 6-20: Streamlines by different turbulence models (Mesh US11)

(a) SST
Figure 6-21: Comparison of pressure coefficient on the airfoil surface of different turbulence models
(a) SST

(b) AKN

(c) SA
Figure 6-22: Comparison of velocity profiles on different locations of different turbulence models

Table 6-13: Cycles to convergence and run times for different turbulence models

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Elements</th>
<th>Cycle</th>
<th>Running time, h:m:s</th>
<th>Run Platform</th>
<th>y+</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST</td>
<td>780,261</td>
<td>4,589</td>
<td>02:44:40</td>
<td>2</td>
<td>11.40</td>
</tr>
<tr>
<td>AKN</td>
<td>780,261</td>
<td>3,651</td>
<td>02:54:27</td>
<td>2</td>
<td>13.20</td>
</tr>
<tr>
<td>SA</td>
<td>780,261</td>
<td>11,472</td>
<td>07:51:40</td>
<td>2</td>
<td>11.54</td>
</tr>
<tr>
<td>SKE</td>
<td>780,261</td>
<td>6,139</td>
<td>03:26:49</td>
<td>2</td>
<td>13.26</td>
</tr>
</tbody>
</table>

Table 6-14: Comparisons of the peak (negative) value of Cp with CFL3D results for different turbulence models. (Note that SST, AKN and SKE results are compared to CFL3D SST results and SA result is compared to CFL3D SA result)

<table>
<thead>
<tr>
<th></th>
<th>SST</th>
<th>AKN</th>
<th>SA</th>
<th>SKE</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFL3D (SST/SA)</td>
<td>-6.8947</td>
<td>-6.8947</td>
<td>-7.3891</td>
<td>-6.8947</td>
</tr>
<tr>
<td>Difference</td>
<td>+4.42%</td>
<td>+8.65%</td>
<td>+2.72%</td>
<td>+5.19%</td>
</tr>
</tbody>
</table>
Additional analysis [3] compares the accuracy of different turbulence models. A key goal is to see which model(s) can resolve the flow separation near the trailing edge. Flow separation can lead to increased pressure drag due to the resulting vortices and a larger pressure differential between the front and rear surfaces of the airfoil as it travels through the fluid. Based on the high angle-of-attack (AoA) of 13.87 degrees, we expect a large region of flow separation near the trailing edge. This AoA is close to the critical value of around 15 degrees. As the AoA increases, the separation point on the upper surface separation moves upstream along the trailing edge toward the leading edge, and the separation region grows. The streamlines show that the SST model reproduces the flow separation well. The SA model also predicts flow separation, but the SKE and AKN models do not. The SST and SA models give the best results for the peak Cp, with a value within 4.42% and 2.72% of the CFL3D (SST and SA) peak. SST and SA accurately predict Cp on the top and bottom surfaces. The velocity profiles of SST and SA are reasonably accurate, but SKE and AKN give very poor velocity profiles. Overall, the SST model gives the best results for additional analysis [3].

6.9 Comparing Structured and Unstructured Mesh Results

It is somewhat difficult to compare the structured and unstructured mesh results since the structured meshes have lower y+ values than all of the unstructured meshes. The maximum y+ for a structured mesh is 2.18, while the minimum y+ for an unstructured mesh is 5.81. From the results, the most accurate predictions of the peak Cp are the structured meshes, however. The finest structured mesh gives a peak within 0.12% of the CFL3D value, while a number of unstructured meshes (with y+ values on the order of 6 – 11) give peaks within 6% of CFL3D. In particular, the US15 mesh (y+ = 11.29) gave an error of just 0.83%. This is despite the fact
that the generally recommended range for the SST turbulence model is $y^+ < 3$. Three of the five structured meshes presented convergence problems, but this did not greatly affect the accuracy of those simulations. When the oscillating predictions were averaged, the results compared well to CFL3D. The unstructured meshes had no convergence problems when the minimum octant size was 1 mm or less. The convergence problems of the structured meshes could be due to elements with high aspect ratios, particularly close to the airfoil. Overall, the velocity profiles for the structured and unstructured meshes were in good agreement with CFL3D. The SST turbulence model gave generally good results for both structured and unstructured meshes. The SA model was also tested with one structured mesh and gave similar accuracy to SST. The SA, AKN and SKE models were tested with one unstructured mesh. SA again did reasonably well, but did not resolve the flow separation region as well as SST. The AKN and SKE models could not capture any flow separation and gave very poor velocity profiles.
7 Conclusions

A numerical study concentrated on four fundamental cases. Structured meshes were tested for all cases, with unstructured meshes also tested for two airfoil cases. It has been demonstrated that as the mesh refinement increases, the $y^+$ values decreases, providing better results for skin friction coefficient, pressure coefficient, and velocity profiles on specific surfaces or at certain locations. Grid independent solutions were obtained for both the SST and SA turbulence models. First, the SST turbulence model was tested with the “SSTD LORE 0” command, which turns off the Wilcox low Reynolds number correction. The test results with the command show better accuracy than those without the command, especially for the flat plate and axisymmetric boundary layer cases. Thus, it is strongly recommended that the Wilcox correction be turned off for the SST turbulence model.

Furthermore, the command “SASW 1” was applied for the SA turbulence model to return it to the standard version. In general, the command had little effect on the results. For the flat plate case, however, the command eliminated transition region problems for coarse meshes, giving much improved profiles of skin friction. For some cases, therefore, using “SASW 1” is necessary to get good results. Different inflow turbulence properties were also tested, but this had no significant effect on the results.

A comparison of structured and unstructured mesh results was attempted for the NACA 0012 and NACA 4412 airfoil cases. The $y^+$ values for the unstructured meshes were higher than those for the structured meshes, however, so it was not possible to obtain a representative comparison of equivalent results. Despite this, it was demonstrated that the unstructured meshes provide an accurate result with little difference from the results of the NASA CFL3D
code. Meanwhile, the AKN and SKE turbulence models were also tested for the NACA 4412 case. Unfortunately, it was found that these two turbulence models could not reproduce the expected flow separation near the trailing edge.

In SC/Tetra steady-state simulations, flow field calculations are repeated until the changes in the field variables from one cycle to the next become so small that the solution can be considered to have converged. When the residuals reach the convergence target, it does not automatically mean that the solution is accurate, however. Conversely, high residuals do not automatically indicate an inaccurate solution. In this work, a convergence target of 1e-6 was used for three of the four cases, but a target of 1e-8 was used for the NACA 0012 case. Unfortunately, most of the NACA 0012 simulations did not converge. Overall, there is no clear evidence that a particular convergence target for the residuals will guarantee more accurate results. It is, therefore, recommended that future testing looks further into how to select a convergence target that is low enough that it will not restrict the accuracy of the results. Or, in other words, a convergence target that will give results that are not influenced by the value of the convergence target itself.

In summary, based on the simulation results for four NASA test cases, the accuracy of the SC/Tetra simulation code has been validated.
8 Reference


Wadcock, A. J. 1979 Structure of the turbulent separated flow around a stalled airfoil. NASA-CR-152263


Abe, N.; Kondoh. 1992 A k-ε model designed with application to turbulence fields accompanied by separation and reattachment. *In Japan Society of Mechanical Engineers Collected Articles, Series B*, **58**, 3003-3010.
