CLEAN WING AIRFRAME NOISE MODELING FOR MULTIDISCIPLINARY DESIGN AND OPTIMIZATION

by Serhat Hosder

Dissertation submitted to the Faculty of the
Virginia Polytechnic Institute and State University
in partial fulfillment of the requirements for the degree of

Doctor of Philosophy
in
Aerospace Engineering

Dr. Bernard Grossman, Chair
Dr. Joseph Schetz, Committee Member
Dr. Raphael T. Haftka, Committee Member
Dr. William H. Mason, Committee Member
Dr. Roger Simpson, Committee Member
Dr. Reece Neel, Committee Member

July 29, 2004
Blacksburg, Virginia

Keywords: Airframe Noise, Aeroacoustics, Trailing Edge Noise, Computational Fluid Dynamics, Multidisciplinary Design and Optimization

Copyright ©2004, Serhat Hosder
Clean Wing Airframe Noise Modeling for Multidisciplinary Design and Optimization

Serhat Hosder

(ABSTRACT)

A new noise metric has been developed that may be used for optimization problems involving aerodynamic noise from a clean wing. The modeling approach uses a classical trailing edge noise theory as the starting point. The final form of the noise metric includes characteristic velocity and length scales that are obtained from three-dimensional, steady, RANS simulations with a two equation $k-\omega$ turbulence model. The noise metric is not the absolute value of the noise intensity, but an accurate relative noise measure as shown in the validation studies. One of the unique features of the new noise metric is the modeling of the length scale, which is directly related to the turbulent structure of the flow at the trailing edge. The proposed noise metric model has been formulated so that it can capture the effect of different design variables on the clean wing airframe noise such as the aircraft speed, lift coefficient, and wing geometry. It can also capture three dimensional effects which become important at high lift coefficients, since the characteristic velocity and the length scales are allowed to vary along the span of the wing.

Noise metric validation was performed with seven test cases that were selected from a two-dimensional NACA 0012 experimental database. The agreement between the experiment and the predictions obtained with the new noise metric was very good at various speeds, angles of attack, and Reynolds Number, which showed that the noise metric is capable of capturing the variations in the trailing edge noise as a relative noise measure when different flow conditions and parameters are changed.

Parametric studies were performed to investigate the effect of different design variables on the noise metric. Two-dimensional parametric studies were done using two symmetric NACA four-digit airfoils (NACA 0012 and NACA 0009) and two supercritical (SC(2)-0710 and SC(2)-0714) airfoils. The three-dimensional studies were performed with two versions of a conventional transport wing at realistic approach conditions. The twist distribution of the baseline wing was changed to obtain a modified wing which was used to investigate the effect of the twist on the trailing edge noise.
An example study with NACA 0012 and NACA 0009 airfoils demonstrated a reduction in the trailing edge noise by decreasing the thickness ratio and the lift coefficient, while increasing the chord length to keep the same lift at a constant speed. Both two- and three-dimensional studies demonstrated that the trailing edge noise remains almost constant at low lift coefficients and gets larger at higher lift values. The increase in the noise metric can be dramatic when there is separation on the wing. Three-dimensional effects observed in the wing cases indicate the importance of calculating the noise metric with a characteristic velocity and length scale that vary along the span. The twist change does not have a significant effect on the noise at low lift coefficients, however it may give significant noise reduction at higher lift values.

The results obtained in this study show the importance of the lift coefficient, $C_L$, on the airframe noise of a clean wing and favors having a larger wing area to reduce the $C_L$ for minimizing the noise. The results also point to the fact that the noise reduction studies should be performed in a multidisciplinary design and optimization framework, since many of the parameters that change the trailing edge noise also affect the other aircraft design requirements. It’s hoped that the noise metric developed here can aid in such multidisciplinary design and optimization studies.
Acknowledgements

First and foremost, I would like to acknowledge my father Mehmet Hosder, my mother Nursin Hosder, and my sister Yasemin Hosder. Without their love and encouragement, I could have never finished this study. They have always believed in me and gave their emotional support at difficult times.

I had the privilege of working with exceptionally good people in my PhD study. Their extensive knowledge, experience, and encouragement played an important role in the success of this project. I am grateful to my advisor, Dr. Bernard Grossman, for his academic advice, support, and encouragement. I would like to thank Dr. Joseph Schetz for his help and guidance. His insight and ideas have played an important role in the success of this work. Dr. Raphael Haftka has always enlightened me with his careful observations, suggestions, and clever ideas. His help is greatly appreciated. I would like to thank Dr. William Mason for his help and advice in every aspect of this study. I was fortunate to benefit from his experience in applied aerodynamics and aircraft design. Dr. Reece Neel was the main source of help for my CFD related questions. He was always very friendly, and I really appreciate his willingness to help me. I would like to thank Dr. Roger Simpson for serving in my thesis committee.

I would like to thank Dr. David Lockard of NASA Langley Research Center for reviewing my thesis and giving helpful comments. It was very kind of him to read a relatively long document in a short amount of time and give useful feedback.

I would like to acknowledge the current and the former student members of the Blended-Wing-Body Design team: Leifur Leifsson, Andy Ko, Vance Dippold, and Jessica Walker. It was a great pleasure to work with them.
I would like to thank Dr. Layne Watson for his participation and help in the work on CFD Simulation Uncertainties.

The CFD code GASP was a crucial element of this study. I am grateful to Aerosoft Inc. for allowing me to use this state of the art CFD software for free.

Financial support for the Airframe Noise Study was supplied by NASA Langley Research Center. The work on CFD Simulation Uncertainties was supported by National Science Foundation.
## Contents

Title Page

Abstract

Acknowledgements

Table of Contents

List of Figures

List of Tables

Nomenclature

1 Introduction

  1.1 Airframe Noise

  1.2 Airframe Noise Prediction

  1.3 Role of CFD in Airframe Noise Prediction

  1.4 Contribution of the Current Study

  1.5 Outline of the Dissertation

2 The Clean Wing Noise Metric
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1 Turbulent Boundary Layer-Trailing Edge Noise</td>
<td>12</td>
</tr>
<tr>
<td>2.2 Derivation of the Noise Metric</td>
<td>15</td>
</tr>
<tr>
<td>2.3 Modeling of $u_0$ and $l_0$</td>
<td>18</td>
</tr>
<tr>
<td>2.4 Lilley’s Clean Aircraft Noise Formulation</td>
<td>19</td>
</tr>
<tr>
<td>2.5 ANOPP Clean Wing Noise Model</td>
<td>21</td>
</tr>
<tr>
<td>2.6 Unique Features of the Proposed Noise Metric</td>
<td>23</td>
</tr>
<tr>
<td>3 CFD Simulations</td>
<td>25</td>
</tr>
<tr>
<td>3.1 Governing Equations</td>
<td>25</td>
</tr>
<tr>
<td>3.2 Numerical Solver</td>
<td>26</td>
</tr>
<tr>
<td>3.3 Physical Modeling</td>
<td>27</td>
</tr>
<tr>
<td>3.3.1 Menter’s $k$-$\omega$ SST Turbulence Model</td>
<td>28</td>
</tr>
<tr>
<td>3.4 Computational Grids</td>
<td>29</td>
</tr>
<tr>
<td>3.4.1 Two-Dimensional Grids</td>
<td>29</td>
</tr>
<tr>
<td>3.4.2 Three-Dimensional Grids</td>
<td>30</td>
</tr>
<tr>
<td>4 Noise Metric Validation</td>
<td>35</td>
</tr>
<tr>
<td>4.1 Description of the Experimental Data</td>
<td>35</td>
</tr>
<tr>
<td>4.2 Outline of the Selected Experimental Test Cases</td>
<td>37</td>
</tr>
<tr>
<td>4.2.1 Semi-Empirical Airfoil Noise Prediction</td>
<td>38</td>
</tr>
<tr>
<td>4.2.2 Calculation of the Overall Sound Pressure Level</td>
<td>41</td>
</tr>
<tr>
<td>4.3 Noise Metric Calculation</td>
<td>42</td>
</tr>
<tr>
<td>4.4 Validation Results</td>
<td>43</td>
</tr>
<tr>
<td>5 Two-Dimensional Parametric Noise Metric Studies</td>
<td>46</td>
</tr>
<tr>
<td>5.1 Need for Parametric Studies</td>
<td>46</td>
</tr>
</tbody>
</table>
5.2 Outline of the Two-Dimensional Studies ........................................ 47
5.3 Studies with NACA 0012 and NACA 0009
   Airfoils .................................................................................... 48
   5.3.1 Geometry Description ....................................................... 48
   5.3.2 Test Cases ......................................................................... 49
   5.3.3 Effect of $C_l$ and $t/c$ on the Noise Metric ....................... 51
   5.3.4 Noise Reduction with $C_l$ and $t/c$ change ......................... 53
5.4 Studies with SC(2)-0710 and SC(2)-0714 Airfoils ......................... 55
   5.4.1 Geometry Description ....................................................... 55
   5.4.2 Test Cases ......................................................................... 57
   5.4.3 Characteristic Velocity and Length Scales ......................... 58
   5.4.4 Noise Metric Predictions .................................................. 64

6 Three-Dimensional Parametric Noise Metric Studies ....................... 69
   6.1 Description of the Baseline Wing Geometry ............................. 70
   6.2 Test Conditions .................................................................... 74
   6.3 Baseline Wing Results ......................................................... 74
      6.3.1 Lift and Drag Characteristics .......................................... 74
      6.3.2 Characteristic Velocity and Length Scale Results ............. 79
      6.3.3 Noise Metric Results ..................................................... 84
   6.4 Effect of the Twist on Noise ................................................ 86
      6.4.1 Modified Twist Distribution ........................................... 87
      6.4.2 Comparison with the Baseline Wing Results ................... 91

7 Discussion and Conclusions ............................................................ 101
   7.1 Summary of the Results ....................................................... 101
List of Figures

1.1 Main components of the aircraft noise. .............................. 2
1.2 The three noise certification reference positions. ............. 3
1.3 Airframe Noise Components. ........................................... 4

2.1 The noise generated due the passage of the turbulent boundary layer over the trailing edge of an airfoil placed in a uniform free-stream flow ...................................................... 12
2.2 The general outline of the Noise Metric derivation ........... 16
2.3 Directivity angles used in the Noise Metric (note that the trailing edge sweep angle (β) is 0° in this figure) .................. 17

3.1 The iteration history of the lift coefficient for a wing case at the fine grid level (Converged $C_L = 0.574$) ...................... 28
3.2 The C-grid topology used in the two-dimensional airfoil cases. 30
3.3 The $z^+$ values on the upper surface the SC(2)-0714 airfoil at $C_L = 1.665$. ................................................................. 31
3.4 The grid around the NACA 0012 airfoil ($c = 0.3048\ m$) used in the CFD simulation of the validation cases. A close-up view of the trailing edge region is given on the right. Every other grid line in the streamwise direction is shown. ................. 32
3.5 The grid around the SC(2)-0714 airfoil \((c = 9.54 \text{ m})\) used in the two-dimensional parametric Noise Metric studies. A close-up view of the trailing edge region is given on the right. Every other grid line in the streamwise direction is shown. .............. 32

3.6 A view of the original wing and the C grid around the root section. 33

3.7 A view of the grid in the wing tip region. .................. 33

3.8 The planform view of the original wing grid used in the three-dimensional Noise Metric studies. .................. 34

4.1 The 1/3-octave Sound Pressure Levels for validation case 7 \((\alpha^* = 1.5^\circ, V_\infty = 71.3 \text{ m/s, and } Re_c = 1.497 \times 10^6)\). Different components of the trailing edge noise spectra are calculated with the airfoil noise prediction method of Brooks et al.\(^28\) .............. 40

4.2 The turbulent kinetic energy and the length scale distributions at the upper surface trailing edge of NACA 0012 airfoil for validation Case 7 \((\alpha^* = 1.5^\circ, V_\infty = 71.3 \text{ m/s, } c = 0.3048 \text{ m and } Re_c = 1.497 \times 10^6)\). .................................................. 43

4.3 The steps followed in the Noise Metric validation study. ...... 44

4.4 The comparison of the Noise Metric predictions \((\overline{NM}_{si})\) and the \(\overline{OASPL}_{si}\) values obtained with ANOPP to the experimental \(\overline{OASPL}_{si}\) values of Brooks et al.\(^28\) at each NACA 0012 validation case. .......................................................... 45

5.1 The NACA 0012 and NACA 0009 airfoils. The airfoil coordinates are made non-dimensional with the chord length \((c)\) and are not to scale. .......................................................... 48

5.2 The section lift coefficient \((C_l)\) vs. the angle of attack \((\alpha)\) and the drag polars obtained for the NACA 0012 and NACA 0009 airfoils. .......................................................... 50
5.3 The total Noise Metric values obtained with NACA 0012 and NACA 0009 airfoils at different section lift coefficients ($Re_c = 1.497 \times 10^6$ and $Mach = 0.20$). 

5.4 The characteristic turbulent velocity ($u_0$) obtained for the suction side of NACA 0012 and NACA 0009 airfoils at different section lift coefficients ($Re_c = 1.497 \times 10^6$ and $Mach = 0.20$).

5.5 The characteristic length scale $l_0$ obtained for the suction side of NACA 0012 and NACA 0009 airfoils at different section lift coefficients ($Re_c = 1.497 \times 10^6$ and $Mach = 0.20$).

5.6 The NACA 0012 and NACA 0009 airfoils with different chord lengths used in the noise reduction study.

5.7 Noise metric reduction history obtained with NACA 0012 and NACA 0009 airfoils for various lift coefficients at constant lift.

5.8 The SC(2)-0710 and SC(2)-0714 airfoils. The airfoil coordinates are made non-dimensional with the chord length ($c$) and are not to scale.

5.9 The drag polars for SC(2)-0710 and SC(2)-0714 airfoils.

5.10 Characteristic turbulent velocity ($u_0$) obtained at the trailing edge of SC(2)-0710 and SC(2)-0714 airfoils at different section lift coefficients ($Re_c = 44 \times 10^6$ and $Mach = 0.20$).

5.11 Characteristic length scale ($l_0$) obtained at the trailing edge of SC(2)-0710 and SC(2)-0714 airfoils at different section lift coefficients ($Re_c = 44 \times 10^6$ and $Mach = 0.20$).

5.12 The skin friction ($C_f$) values obtained at the trailing edge of the suction side of SC(2)-0710 and SC(2)-0714 airfoils for different section lift coefficients.
5.13 The turbulent kinetic energy (TKE) and the length scale \( l_0 \) profiles at the upper surface trailing edge of SC(2)-0710 and SC(2)-0714 airfoils for various section lift coefficients. The filled symbols show the maximum TKE and the corresponding length scale values.

5.14 Velocity profiles at the trailing edge of the SC(2)-0714 airfoil at different section lift coefficients. The zero pressure gradient case is calculated with theoretical predictions\(^\text{58}\) at \( Re_c = 44 \times 10^6 \) and shown only for qualitative comparison. Red dashed-line shows the \( TKE_{max} \) location for \( C_l = 0.550 \), and black dashed-line marks the \( TKE_{max} \) location for \( C_l = 1.665 \).

5.15 The Noise Metric values obtained for the suction and pressure sides of the SC(2)-0710 airfoil. The total Noise Metric value is obtained using Equation 2.11.

5.16 The Noise Metric values obtained for the suction and pressure sides of the SC(2)-0714 airfoil. The total Noise Metric value is obtained using Equation 2.11.

5.17 The Comparison of total Noise Metric values obtained with SC(2)-0710 and SC(2)-0714 airfoils.

5.18 Comparison between the scaled total Noise Metric value \((\text{NM}_s)\) of SC(2)-0714 airfoil and the scaled Overall Sound Pressure Level \((\text{OASPL}_s)\) obtained with the formula by Lockard and Lilley\(^{10}\) (Equation 2.17), and ANOPP.\(^{27}\)

6.1 EET Wing Planform

6.2 The airfoils used in the root, break, and the tip location of the EET wing. The airfoil coordinates are made non-dimensional with the local chord length \((c)\) and are not to scale.

6.3 The spanwise variation of the maximum thickness ratio \((t/c)\) and the actual maximum thickness \((t)\).

6.4 The baseline twist distribution \((\theta_b)\) of the baseline wing.
6.5 Overall lift coefficient \( (C_L) \) and Lift Loading \( (L/S_{ref}) \) vs. angle of attack \( (\alpha) \) for the baseline wing. ........................................ 76

6.6 The drag polar of the baseline wing. ........................................ 76

6.7 Section lift coefficient \( (C_l) \) distributions for the baseline wing. . 78

6.8 Spanload distributions for the baseline wing. ............................ 78

6.9 Maximum section lift coefficient \( (C_{lmax}) \) and its ratio to the overall lift coefficient \( (C_{lmax}/C_L) \) for various \( C_L \) obtained with the baseline wing. ........................................ 79

6.10 Skin friction contours on the upper surface of the baseline wing at different \( C_L \) values. ................................................................. 80

6.11 Maximum \( TKE \) \( (u_o^2) \) distributions along the upper surface trailing edge of the baseline wing. ........................................ 82

6.12 Characteristic length scale \( (l_0) \) distributions along the upper surface trailing edge of the baseline wing. ........................................ 82

6.13 Turbulent Kinetic Energy contours in the vicinity of the baseline wing tip trailing edge region (looking from downstream) at different \( C_L \) values. Note that the maximum TKE of the last case \( (C_L = 1.106) \) is much greater than the contour upper limit. .... 83

6.14 Noise metric values obtained with the baseline wing at different lift coefficient values. In the abscissa, \( C_L \) stands for the lift coefficient calculated based on the wing planform area \( (S_{ref}) \), and \( C_{Ltp} \) is the lift coefficient that uses the trapezoidal wing area \( (S_{tp}) \) as the reference scaling value. ........................................ 85

6.15 The wing planform with vortex lattice locations used in VLMpc code. ................................................................. 88

6.16 The section lift comparison for the baseline wing at \( C_L = 0.970 \). 88

6.17 The baseline and the modified twist distributions. .................... 89

6.18 Section lift coefficient values for the baseline and modified wing at \( C_L = 0.970 \). Results were obtained using the VLMpc code. ... 90
6.19 Comparison between the overall lift coefficient ($C_L$) and the wing loading ($L/S_{ref}$) values of the baseline and the modified wing at various angles of attack. ................................. 92

6.20 The comparison between the drag polars of the baseline and the modified wing. ................................. 93

6.21 The comparison between the total Noise Metric values of the baseline and the modified wing at different lift coefficient values. 94

6.22 Maximum $TKE$ ($u_0^2$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.375$. . . 96

6.23 Characteristic length scale ($l_0$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.375$. ................................. 96

6.24 Maximum $TKE$ ($u_0^2$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.970$. . . 97

6.25 Characteristic length scale ($l_0$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.970$. ................................. 97

6.26 Contributions to the total Noise Metric from the inboard and the outboard sections of the baseline and the modified wing at $C_L = 0.970$. ................................. 99

6.27 Section lift coefficient ($C_l$) distributions for the baseline and the modified wing at $C_L = 0.970$. ................................. 99

6.28 Spanload distributions for the baseline and the modified wing at $C_L = 0.970$. ................................. 100

6.29 The comparison between the scaled total Noise Metric values ($NM_s$) and the scaled Overall Sound Pressure Levels ($OASPL_s$) obtained with ANOPP, and with the formula by Lockard and Lilley (Using both $C_L$ and $C_{lmax}$ in Equation 2.17). ................................. 100
A.1 The trailing edge plane of a wing grid used in three-dimensional parametric studies. The $TKE$ and $\omega$ values are extracted from the cell centers at the trailing edge plane.

A.2 The turbulent kinetic energy ($TKE$) profile at the trailing edge of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The black circles show the $TKE$ values at the cell center locations and black arrow points the maximum value. The red line represents the $2^{nd}$ order polynomial fit to the $TKE$ in the vicinity of the maximum obtained from the cell center values and the red circle is the maximum value ($\tilde{TKE}_{max}$) obtained from the $2^{nd}$ order polynomial fit.

A.3 The turbulence frequency ($\omega$) profile at the trailing edge of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The black circles show the $\omega$ values at the cell center locations and black arrow points the $\omega$ at the maximum $TKE$ location obtained from the cell center values. The red line represents the $2^{nd}$ order polynomial fit to the $\omega$ and the red circle is the $\tilde{\omega}$ value obtained from the fit at the $\tilde{TKE}_{max}$ location.

A.4 The characteristic length scale at the trailing edge of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The black circles show the length scale values at the cell center locations and black arrow points the $l_0$ at the maximum $TKE$ location obtained from the cell center values. The red circle is the $l_0$ value obtained with the numerical procedure described in Appendix A.

A.5 A portion of the length scale ($l_0$) distribution along the span of a wing used in the preliminary noise metric studies. The symbols show the length scale distribution obtained before and after the application of the numerical procedure described in Section A.
B.1 Original geometry, Grid 2 (top), and extended geometry, Grid $2_{\text{ext}}$ (bottom), used in the transonic diffuser computations. The flow is from left to right, in the positive $x$-direction. The $y$-direction is normal to the bottom wall. All dimensions are scaled by the throat height, $h_t$. The throat section, which is the minimum cross-sectional area of the channel, is located at $x/h_t = 0.0$. Both geometries have the inlet stations located at $x/h_t = -4.04$.

B.2 Velocity contours, streamlines, and the top wall pressure distributions of the weak and the strong shock cases.

B.3 Normalized $L^2$ residual of the energy equation for the case with the Sp-Al turbulence model, Van Albada, and Min-Mod limiters at $P_e/P_{0i} = 0.72$ obtained with the original geometry. Normalization is done with the initial value of the residual.

B.4 Convergence history of the nozzle efficiency at different grid levels for the Sp-Al, Min-Mod, strong shock case obtained with the original geometry. (The nozzle efficiency values are monitored at every 50 cycles starting from iteration number 10000 for Grid 5).

B.5 Nozzle efficiencies obtained with different grid levels, turbulence models, limiters, geometries, and boundary conditions for the strong shock case (A) and the weak shock case (B).

B.6 Nozzle efficiency vs. exit pressure ratio for different grids obtained with the original geometry, Sp-Al and $k$-$\omega$ turbulence models, and the Min-Mod limiter.

B.7 Mach number values at the upstream of the shock ($x/h_t = -1.5$), and downstream of the shock ($x/h_t = 8.65$, the exit plane) for different grids obtained with the original geometry, Sp-Al and $k$-$\omega$ turbulence models, Min-Mod and Van Albada limiters. The values of $y/h_t$ correspond to the mid points of the local channel heights.
B.8 Error distribution in $y/h_t$ for the upper wall of the modified-wall diffuser geometry at the data points measured in the experiments. The maximum error is approximately 7% and observed upstream of the throat, at $x/h_t = -1.95$. Starting from $x/h_t = 1.2$, the error is approximately constant with an average value of 0.9%.

B.9 Upper wall contours of the original and the modified-wall diffuser geometry in the vicinity of the throat location.

B.10 Top wall pressure distributions obtained with the original and the modified-wall geometry for the strong shock case (the results of the Sp-Al model, Min-Mod limiter, and Grids $g_2$ and $g_{2_{mw}}$ are shown).

B.11 Top wall pressure distributions obtained with the original and the modified-wall geometry for the weak shock case (the results of the Sp-Al model, Min-Mod limiter, and Grids $g_2$ and $g_{2_{mw}}$ are shown).

B.12 Streamline patterns of the separated flow region obtained with different versions of the diffuser geometry and exit pressure ratios for the strong shock case.

B.13 Comparison of the separation bubbles obtained with different versions of the diffuser geometry and exit pressure ratios for the strong shock case.

B.14 Top wall pressure distributions obtained with different versions of the diffuser geometry and exit pressure ratios for the strong shock case (the results of the Sp-Al model, Van Albada limiter, and Grids $g_3$ and $g_{3_{ext}}$ are shown).
# List of Tables

4.1 Experimental NACA 0012 airfoil test cases used in the Noise Metric Validation. ................................................................. 38

4.2 The Overall Sound Pressure Level and the Noise Metric values obtained for the validation cases. .................................................. 41

5.1 The NACA 0012 and NACA 0009 cases used to study the effect of $C_l$ and $t/c$ on the Noise Metric. For all the cases, $Re_c = 1.497 \times 10^6$, $Mach = 0.2$, $V_{\infty} = 71.3 \, m/s$, $c = 0.3047 \, m$, $\theta = 90^\circ$, $\psi = 90^\circ$, and $H = 1.22 \, m$. .................................................. 49

5.2 The SC(2)-0714 and SC(2)-0710 cases used in the Noise Metric studies. For all the cases, $Re_c = 44 \times 10^6$, $Mach = 0.2$, $V_{\infty} = 68.0 \, m/s$, $c = 9.54 \, m$, $\theta = 90^\circ$, $\psi = 90^\circ$, and $H = 120 \, m$. .................................................. 57

6.1 EET wing geometry parameters ................................................................. 71

6.2 The baseline wing cases of the three-dimensional Noise Metric studies. For all the cases, $Re_c = 44 \times 10^6$, $Mach = 0.2$, $U_{\infty} = 68 \, m/s$, $mac = 9.54 \, m$, $\theta = 90^\circ$, and $H = 120 \, m$. Note that $C_L$ is the lift coefficient calculated based on the wing planform area ($S_{ref}$), and $C_{Ltp}$ is the lift coefficient that uses the trapezoidal wing area ($S_{tp}$) as the reference scaling value. .................................................. 75

6.3 The modified wing cases of the three-dimensional Noise Metric studies. For all the cases, $Re_c = 44 \times 10^6$, $Mach = 0.2$, $U_{\infty} = 68 \, m/s$, $mac = 9.54 \, m$, $\theta = 90^\circ$, and $H = 120 \, m$. Note that $C_L$ is the lift coefficient calculated based on the wing planform area ($S_{ref}$), and $C_{Ltp}$ is the lift coefficient that uses the trapezoidal wing area ($S_{tp}$) as the reference scaling value. .................................................. 91
6.4 Comparison between the drag and the Noise Metric values of the baseline and the modified wings at selected lift coefficients ........................................... 95

B.1 Different versions of the transonic diffuser geometry and exit pressure ratios \((P_e/P_{bi})\) used in the computations. ............................................................. 136

B.2 Mesh size nomenclature for the transonic diffuser case. In the simulations, five different grids were used for the original geometry: Grid 1 (g1), Grid 2 (g2), Grid 3 (g3), Grid 4 (g4), and Grid 5 (g5). The finest mesh is Grid 5 and the other grids are obtained by reducing the number of divisions by a factor of 2 in both \(x\)- and \(y\)-directions at each consecutive level (grid halving). Grid 5 is used only for the case with the Sp-Al turbulence model, Min-Mod limiter, and \(P_e/P_{bi} = 0.72\). Four grid levels were used for the extended geometry: Grid 1_{ext} (g1_{ext}), Grid 2_{ext} (g2_{ext}), Grid 3_{ext} (g3_{ext}), and Grid 4_{ext} (g4_{ext}). The grids for the extended geometry and the grids generated for the original geometry are essentially the same between the inlet station and \(x/h_t = 8.65\). For the modified-wall geometry, three grid levels were used: Grid 1_{mw} (g1_{mw}), Grid 2_{mw} (g2_{mw}), and Grid 3_{mw} (g3_{mw}). All the grids have the same mesh distribution in the \(y\)-direction. ........... 137

B.3 Main observations on uncertainty sources ................................................................. 138

B.4 Discretization error results of the transonic diffuser case obtained with the original geometry. The cases presented in this table exhibit monotonic convergence with the refinement of the mesh size. For each case with a different turbulence model, limiter, and exit pressure ratio, the approximation to the exact value of \(n_{eff}\) is denoted by \((\tilde{n}_{eff})_{exact}\) and the discretization error at a grid level \(k\) is calculated by \(\text{error} \, (\%) = \left| \frac{(n_{eff})_k - (\tilde{n}_{eff})_{exact}}{(\tilde{n}_{eff})_{exact}} \right| \times 100\). 139

B.5 Top wall orthogonal distance error \(\hat{E}_n\) calculated upstream of the experimental shock location (UESL) for each case obtained with the original geometry. Scaled error values \(\hat{E}_n\) were obtained by \(\hat{E}_n = \frac{E_n}{(E_n)_{max}} \times 100\) where \((E_n)_{max}\) is the maximum \(E_n\) value calculated downstream of the experimental shock location (DESL) for the strong shock case with Grid 4, Min-Mod limiter, and the \(k-\omega\) turbulence model. ........... 140
B.6 Top wall orthogonal distance error $\hat{E}_n$ calculated downstream of the experimental shock location (DESL) for each case obtained with the original geometry.

B.7 Nozzle efficiency values obtained with different grid levels, limiters, turbulence models, geometries, and boundary conditions.

B.8 Main observations on the uncertainty in nozzle efficiencies.

B.9 Discretization errors calculated by using the results of different grid levels for the transonic diffuser case with the original geometry, Sp-Al turbulence model, and the Min-Mod limiter.
Nomenclature

Abbreviations
ANOPP = Aircraft Noise Prediction Program
CAA = computational aeroacoustics
CFD = computational fluid dynamics
FAA = Federal Aviation Administration
FAR = Federal Aviation Regulations
ICAO = International Civil Aviation Organization
RANS = Reynolds averaged Navier-Stokes
TBL-TE = turbulent boundary layer-trailing edge
TE = trailing edge
LE = leading edge

Roman Symbols
\( a \) = speed of sound
\( a_\infty \) = free-stream speed of sound
\( b \) = wing span
\( c \) = chord
\( c_a \) = mean geometric chord
\( C_d \) = section drag coefficient
\( C_D \) = overall drag coefficient calculated with \( S_{ref} \)
\( C_{D_{tp}} \) = overall drag coefficient calculated with \( S_{tp} \)
\( C_f \) = skin friction coefficient
\( C_l \) = section lift coefficient
\( C_{l_{max}} \) = maximum section lift coefficient along the span
\( C_{l\alpha} \) = section lift-curve slope
$C_L$ = overall lift coefficient based on $S_{ref}$

$C_{La}$ = overall lift-curve slope

$C_{L0}$ = overall lift coefficient at $\alpha = 0^o$

$C_{Ltp}$ = overall lift coefficient based on $S_{tp}$

$D$ = directivity function

$EPNL$ = Effective Perceived Noise Level

$f$ = frequency

$H$ = distance to the ground or receiver

$I$ = noise intensity

$I_{NM}$ = noise intensity indicator

$k$ = acoustic wave number ($\omega_0/a$)

= turbulent kinetic energy

$L/S_{ref}$ = lift loading

$l_0$ = characteristic length scale for turbulence

$mac$ = mean aerodynamic chord

$M_\infty$ = free-stream Mach number ($V_\infty/a_\infty$)

$MTOW$ = maximum take-off weight of the aircraft

$NM$ = noise metric

$NM_{upper}$ = noise metric for wing upper surface

$NM_{lower}$ = noise metric for wing lower surface

$OASPL$ = overall sound pressure level

$p$ = pressure

$p'$ = acoustic pressure

$q_i$ = heat flux vector component

$Re_{mac}$ = Reynolds number based on $mac$

$Re_c$ = Reynolds number based on chord

$Re_{le}$ = airfoil leading edge radius

$r_0$ = distance from the center of the eddy (quadrupole) to the edge of the half plane

$SPL$ = sound pressure level

$S_{ref}$ = wing planform area

$S_{tp}$ = trapezoidal wing area

$St$ = Strouhal number

$T$ = temperature

$t/c$ = maximum thickness ratio
$T_{ij}$ = Lighthill’s turbulence stress tensor

$TKE$ = turbulent kinetic energy

$TKE_{max}$ = maximum turbulent kinetic energy

$u_i$ = velocity components in cartesian coordinates

$u_0$ = characteristic velocity scale for turbulence

$u_*$ = friction velocity

$V_\infty$ = free-stream velocity

$W$ = weight of the aircraft

$W/S_{ref}$ = wing loading

$x$ = streamwise coordinate

$y$ = spanwise coordinate

$z_n$ = direction normal to wing surface

$(z_n)_{max}$ = location of the maximum turbulent kinetic energy

**Greek Symbols**

$\alpha$ = angle of attack

$\alpha^*$ = effective angle of attack

$\beta$ = trailing edge sweep angle

$\delta$ = boundary layer thickness

$\delta^*$ = displacement thickness

$\nu$ = kinematic viscosity

$\omega$ = turbulence frequency

$\delta_{ij}$ = kronecker delta

$\omega_0$ = characteristic source frequency

$\psi$ = azimuthal directivity angle

$\rho$ = density

$\rho'$ = acoustic density fluctuation

$\rho_\infty$ = free-stream density

$\tau_{ij}$ = viscous shear stress tensor

$\tau_w$ = wall shear stress

$\theta$ = polar directivity angle

$= \text{wing twist angle}$
Chapter 1

Introduction

Noise can be defined as sound that produces adverse affects.\(^1\) With this definition, it is obvious that the aircraft is a major source of noise which can affect people within a certain radius of its path as well as its crew and passengers inside. Aerodynamic noise is generated whenever the passage of air over the aircraft structure or through its power-plants causes fluctuating pressure disturbances that propagate to an observer in the aircraft or on the ground below.\(^1\) The crew and the passengers on the aircraft are exposed to interior or cabin noise.\(^2\) The subject of the current study is the noise received on the ground and created by subsonic civil transport aircraft.

Due to the negative impact on public comfort and health, aircraft noise has become an important performance criterion and constraint in aircraft design in recent years. Although there has been a dramatic reduction in aircraft noise in the last three decades with the advances in airframe and engine technology, further reduction is still needed to allow civil aviation to grow and to minimize noise pollution. Aircraft noise regulations have had the effect of curtailing the growth of air transportation. These regulations limit the hours and the number of operations at most airports and impede aviation infrastructure improvements such as airport expansion and construction plans.\(^3\) There has been almost a 100\% increase in the number of noise related restrictions in the last decade,\(^4\) and the number of airports affected by these noise restrictions has grown significantly worldwide.

The noise related restrictions have an important effect on the design of the new transport aircraft. Boeing\(^5\) is designing its new 7E7 series with the goal of low noise signature
to fulfill the current and future noise requirements set by the civil aviation authorities. Airbus also aims to meet the strict noise regulations with its newest product A380 by the introduction of new generation engines, advanced wing and undercarriage design and technology. Besides the efforts of the aircraft industry, NASA also set the goal of reducing the perceived noise level of the future aircraft by 10 dB from the subsonic aircraft of 1997 within 10 years and by 20 dB in 25 years to tackle the aircraft noise problem and its negative impact on the future of civil aviation. To achieve this challenging noise reduction goal, research efforts have been focused on: (1) the design of revolutionary aircraft with innovative configurations and technologies to give the minimum noise signature (2) the improvement of the noise performance of conventional aircraft, and (3) optimizing the flight performance parameters or operational conditions for minimum noise. All these efforts clearly require addressing noise in the aircraft conceptual design phase.

To include aircraft noise as a constraint or an objective function in a Multidisciplinary Design and Optimization (MDO) framework, each noise component must be modeled. These models are required to predict the aircraft noise originating from different sources in different flight regimes.

The engine, airframe, and the interference between the engine and airframe are the main sources of the aircraft noise (Figure 1.1). Each source consists of sub-components that contribute to the overall noise level. The noise radiating from each source covers a different fraction of the total noise at different flight regimes. In particular, one is interested in the noise signature of a civil transport aircraft at three specific points as shown in Figure 1.2. These are the three noise certification reference points for civil transport aircraft set by FAA in FAR Part 36 and ICAO in Annex 16. The certification
Figure 1.2: The three noise certification reference positions.

requires the Effective Perceived Noise Level (EPNL) of the aircraft to be less than a maximum allowable level at each location. The maximum allowable level changes depending on the Aircraft Maximum Take-off Weight (MTOW). During acceleration on the ground and at take-off, the dominant noise source is the engine. However, the use of high-bypass ratio turbofan engines and other achievements in engine technology make the airframe noise level of conventional transport aircraft comparable to the engine noise on the approach.\(^1\) This brings out the fact that any further reduction of aircraft noise on the approach can only be achieved if both engine and airframe noise are reduced by roughly equal amounts.\(^10\)

The current study focuses on the airframe noise on approach. Therefore, the noise studies presented in this work will be performed with the conditions at the approach reference point (Figure 1.2), which corresponds to the position of an aircraft on a 3° glide slope, approximately 2000 m before the touchdown at an altitude of 120 m.

### 1.1 Airframe Noise

Airframe noise is defined as the *non-propulsive* noise of an aircraft in flight.\(^11\) Airframe noise sources on a conventional transport are the landing gear, trailing edge flaps, leading edge slats, the clean wing, and tail surfaces\(^12\) (Figure 1.3). A clean wing (or clean aircraft) is defined as the configuration that has all high-lift devices and the undercarriage in stowed positions.

The flap noise originates from the flap trailing edges and flap side edges. Recent experi-
Airframe Noise Sources

Figure 1.3: Airframe Noise Components.

ments by Guo et al.\textsuperscript{13,14,15} and Stoker\textsuperscript{16} show that the flap side edge is the main region that dominates the flap noise. The strong roll-up vortex formed due to the sharp change in lift between the flapped and the unflapped portion of the wing is responsible for the flap side edge noise.\textsuperscript{10} In the vicinity of the flap side edges, separated flow regions contain high turbulence and pressure fluctuations which increase the noise level in significant amounts.

One of the major sources of the airframe noise is the unsteady flow in the leading edge slat region of the high-lift system.\textsuperscript{17} The unsteady flow in the slat region of a high lift system is generally very complex and dominated by the viscous effects. Lockard and Lilley\textsuperscript{10} define the mechanism for the high-frequency tonal slat noise as the resonance between the vortex shedding from the trailing edge of the slat and the gap between slat and the main wing section. They also address the instabilities in the slat cove shear layer, which produce the broadband component of the slat noise.

The landing gear is the dominant airframe noise on approach.\textsuperscript{18} The noise source is the turbulent, unsteady, separated flow around various components of the landing gear. Since the landing gear has many cavities and sharp edges, the flow-field is very complex with three-dimensional separation regions of different sizes. The landing gear far-field noise varies with approximately the sixth power of the aircraft speed.\textsuperscript{1}

The main noise mechanism of a clean wing is the Trailing Edge (TE) Noise. The trailing edge noise originates from the scattering of the acoustic waves generated due to the passage of the turbulent boundary layers over the trailing edges of wings or flaps. The
experiments\textsuperscript{19} and different theories\textsuperscript{20,11} on the trailing edge noise demonstrate that the far-field noise intensity varies approximately with the $5^{th}$ power of the free-stream velocity.

Experiments\textsuperscript{21} and Flight measurements\textsuperscript{22} on the airframe noise show that the landing gear, flap side edges, and the leading edge slats are the dominant noise sources for a typical transport aircraft on approach. The deployment of the high-lift devices and the landing gears can increase the overall airframe noise level of the clean aircraft (wing) by about 10 $dB$\textsuperscript{1}. The turbulent boundary layer scattering from a wing trailing edge does not contribute to the total airframe noise as much as the high-lift devices and the landing gear. However,

- Trailing Edge Noise can be a significant contributor to the airframe noise for a non-conventional configuration that does not use traditional high-lift devices on approach such as a Blended-Wing-Body (BWB) transport aircraft, which has a large wing area and span, a conventional aircraft or a BWB with distributed propulsion\textsuperscript{23,24} that uses the jet-wing concept for high-lift, or an airplane with a morphing wing.

- A Trailing Edge Noise formulation based on proper physics may also be used to predict the noise originating from the flap trailing-edges at high lift conditions.

- Trailing Edge Noise of a conventional wing at high lift can be thought as a lower bound value of the airframe noise on approach as defined by Lockard and Lilley\textsuperscript{10}. In other words, if the same lift required on approach can be obtained without using the traditional high-lift devices, the noise of the clean wing would be the lowest value that can be achieved for that particular aircraft as long as there is no massive separation on the wing. This value can be used as a measure of merit in noise reduction studies.

- Lockard and Lilley\textsuperscript{10} also show that even if all noise from the landing gear and high-lift system are eliminated, the NASA goal of 10 $dB$ reduction will still not be met. In other words, the lower bound value for a conventional transport aircraft is still going to be larger than the target noise level. This implies that trailing edge noise must also be reduced to achieve the NASA goal.

With the motivation of the above facts, and as the first step towards a general MDO
noise model, the current study has focused on airframe noise modeling of a clean wing at approach conditions.

1.2 Airframe Noise Prediction

Most of the airframe noise prediction methods used in aircraft design\textsuperscript{4,25} and analysis are based on semi-empirical relations. Among these, the most widely used method is the one developed by Fink,\textsuperscript{26} which is based on data from experiments and flight measurements performed in the 1970’s. NASA Langley’s Aircraft Noise Prediction Program\textsuperscript{27} (ANOPP) uses Fink’s Method in its airframe noise prediction module. In ANOPP, airframe noise sources include the clean wing, tail, landing gear, flaps and leading edge slats. Broadband noise for each component is calculated using Fink’s methodology, which consists of empirical functions to produce sound spectra as a function of frequency, polar directivity angle, and azimuthal directivity angle. Guo \textit{et al.}\textsuperscript{14} have recently developed an empirical model for predicting noise from high-lift systems. They derived the model from a large database of airframe noise sets, involving various airplane models at various operating conditions. Their model correlates noise to gross airplane parameters such as the dimensions of the high-lift system and the Mach number and also to flow quantities that are physically responsible for the noise generation such as the side-edge vortex strength and the crossflow velocity in the case of calculating the flap side edge noise. Brooks \textit{et al.}\textsuperscript{28} performed several experiments with NACA 0012 airfoils having different chord lengths, at different angles of attack and different free-stream velocities. They investigated different noise source mechanisms including the turbulent boundary layer-trailing edge noise. Their data from the experiments included the Sound Pressure Level (SPL) spectra of different noise sources. They also used this data set to develop a semi-empirical airfoil self-noise prediction method. Some of the test cases from their experimental study are used in the validation of the method developed in this study (See Chapter 4).

1.3 Role of CFD in Airframe Noise Prediction

In recent years, Computational Fluid Dynamics (CFD) has actively been used in the airframe noise prediction. Computational Fluid Dynamics is an inherent part of the
Computational Aeroacoustics (CAA). Wells and Renaut\(^{29}\) give an overview of calculating aerodynamically generated noise using CAA methods. Most of the CAA methods used today in airframe noise prediction utilize a hybrid strategy. In these methods, the first step consists of calculating the unsteady flow field in the noise source region. The second part deals with the calculation of the noise in the acoustic far-field. The unsteady near flow field calculated in the first step is the input for the second part. Most of the acoustic codes used in the second step are developed based on the Ffowcs Williams and Hawkings\(^{30}\) equation, which is the most general form of Lighthill’s acoustic analogy.\(^{31},^{32}\) It should be noted that the computed unsteady flow-field used as an input to the acoustic code should be highly accurate in order to calculate the correct far-field noise. This leads to the requirement that the unsteady flow simulations should be performed with high fidelity CFD tools on very fine grids. Although direct numerical simulation (DNS) or large eddy simulation methods (LES) are used in aeroacoustic study of simple problems,\(^{33}\) time-accurate Reynolds Averaged Navier-Stokes (RANS) solvers are the common CFD tools used today to provide the unsteady flow field information around realistic airframe noise components such as the flaps, slats, or the landing gear. Singer et al.\(^{34}\) performed computational simulations of acoustic scattering from a trailing edge, where the radiated noise has been computed using a time-accurate RANS solver coupled to Lighthill’s Acoustic Analogy\(^{31},^{32}\) in the form presented by Ffowcs Williams and Hawkings.\(^{30}\) Khorrami et al.\(^{35}\) used the same approach for time-accurate simulations and acoustic analysis of a slat free-shear layer. Lockard et al.\(^{18}\) calculated the unsteady flow field around a simplified landing gear with 13 million grid points and used the Ffowcs Williams and Hawkings equation to predict the noise at the far-field.

The Computational Aeroacoustics methods can give accurate results, however they are very costly due to the computational expense associated with the very fine time and space resolution requirements. For an MDO problem involving aerodynamic noise from a clean wing, considering the number of runs to be performed for creating response surfaces, it is impractical to use Computational Aeroacoustics. Steady, RANS simulations may supply useful information about the mean flow structure which can be used in models for noise prediction. In fact, the current study includes such an approach which uses steady, three-dimensional RANS simulations with a two-equation turbulence model to calculate the characteristic velocity and the length scales used in the noise prediction model developed. Also, with today’s computers and efficient parallel algorithms, using
steady RANS simulations in design studies is no longer prohibitive.

As the importance of the CFD increases as a design and analysis tool in noise prediction as well as in other fields, the accuracy of the solutions obtained with the CFD simulations becomes more of a concern for the analyst or the designer especially when the flow problem is complex. This raises the need to understand the sources of CFD simulation errors and their magnitudes to be able to assess the magnitude of the uncertainty in the results. For the interested reader, Appendix B presents a study, which illustrates different sources of uncertainty in CFD simulations by a careful study of a typical, but complex fluid dynamics problem. In this study, the uncertainty in CFD simulation results has been studied in terms of five contributions: (1) iterative convergence error, (2) discretization error, (3) error in geometry representation, (4) turbulence model, and (5) changing the downstream boundary condition. The magnitudes and importance of each source of uncertainty is compared. The study presented in Appendix B provides detailed information about the sources and magnitudes of uncertainties associated with the numerical simulation of complex flow fields.

1.4 Contribution of the Current Study

In this study, a new Noise Metric has been developed for constructing response surfaces that may be used for optimization problems involving aerodynamic noise from a clean wing. The noise metric is not the absolute value of the noise intensity, but an accurate relative noise measure as shown in the validation studies. The modeling approach uses the theory of Ffowcs Williams and Hall\textsuperscript{36} as the starting point. The final form of the noise metric includes characteristic velocity and length scales that are obtained from three-dimensional, steady, RANS simulations with a two equation turbulence model. One unique feature of the noise metric is the modeling of the length scale which is believed to be a better indicator of the turbulence structure at the wing trailing edge compared to the other quantities suggested in the literature such as the boundary layer or the displacement thickness. The noise model is also capable of capturing three dimensional effects which become important at high lift coefficients.

Many of the clean wing noise prediction methods used today are based on semi-empirical relations. The empirical nature of these methods may limit the accuracy level of their
predictions when the problem variables (flow conditions, geometries, etc.) are different than the range of parameters used for building the empirical database. One of the benefits of the new noise metric approach is to be able model characteristic velocity and length scale by using RANS solutions to achieve better noise prediction for different flow conditions and geometries.

The noise metric is developed so that it can capture the effects of different design variables on the clean wing airframe noise such as the aircraft speed, lift coefficient and wing geometry (thickness ratio $t/c$, airfoil shape, twist, trailing edge sweep, etc). These variables vary the characteristic velocity and the length scale which are obtained from the RANS simulations. Most empirical noise prediction methods ignore the effect of such parameters on the velocity and the length scale.

This study also includes two- and three-dimensional parametric studies which investigate the effect of wing geometry and the lift coefficient on the clean wing airframe noise. The information obtained from these studies not only contributes to the general knowledge in the field, but also helps the selection of the appropriate design parameters that may be used in optimization problems involving aerodynamic noise from a clean wing.

1.5 Outline of the Dissertation

The details about the new trailing edge noise metric developed in the current study are presented in Chapter 2. This chapter first gives a brief review of the turbulent boundary layer-trailing edge noise theory. Following the review, the derivation of the new noise metric is described. The modeling of the characteristic velocity and the length scales that are used in the noise metric is described next. Chapter 2 ends with a presentation of the unique features of the new noise metric. Chapter 3 gives a review of the governing equations and the physical models used in the CFD simulations. The description of the computational grids used in the noise metric studies are also included in this chapter. Then, Chapter 4 gives the noise metric validation studies. The experimental test cases and the corresponding CFD simulations used in the noise metric validation are described. Following the descriptions, the validation strategy and the results of the validation are presented. Chapter 5 presents two-dimensional parametric noise metric studies, which were performed with two symmetric NACA four-digit airfoils and two
supercritical airfoils. The effect of the thickness ratio and the section lift coefficient on the trailing edge noise is studied with great detail in this chapter. The three-dimensional parametric noise metric studies performed with two versions of a conventional transport aircraft wing are then presented in Chapter 6. The effect of the overall lift coefficient and the twist is explained here. The changes in the noise due to three-dimensional effects are also demonstrated in this chapter. The results are summarized in Chapter 7 which ends with a discussion about the implications of these results on design studies involving aerodynamic noise from a clean wing. Appendix A gives an explanation of the method used to extract the characteristics velocity and length scales from the RANS simulations. Lastly, Appendix B presents a detailed study on CFD simulation uncertainties.
Chapter 2

The Clean Wing Noise Metric

The details about the trailing edge Noise Metric used in the current study are presented in this chapter. First, a brief review of the turbulent boundary layer-trailing edge noise theory is given. Following the review, the derivation of the new Noise Metric is described. Then, the modeling of the characteristic velocity and the length scales that are used in the Noise Metric is described. A detailed explanation of the method used to extract these characteristics scales from the RANS simulations can be found in Appendix A. A brief description of clean aircraft noise prediction formulation by Lockard and Lilley\textsuperscript{10} is given next. Both the current Noise Metric approach and the formulation by Lockard and Lilley\textsuperscript{10} use theoretical results of Ffowcs Williams and Hall\textsuperscript{36} as the starting point in their derivation, however there are differences between the methodologies, especially in the modeling of the characteristic velocity and the length scales. These differences are also stated in the same section. Following this section, the clean wing airframe noise model used in NASA’s Aircraft Noise Prediction Program\textsuperscript{27} (ANOPP) is described briefly. Clean wing noise predictions from ANOPP and the model by Lockard and Lilley\textsuperscript{10} are compared to the predictions of the new Noise Metric developed here for selected two- and three-dimensional cases in the parametric trailing edge noise studies (See Chapters 5 and 6). Finally, this chapter ends with a presentation of the unique features of the trailing edge Noise Metric developed in the current study.
Figure 2.1: The noise generated due the passage of the turbulent boundary layer over the trailing edge of an airfoil placed in a uniform free-stream flow

2.1 Turbulent Boundary Layer-Trailing Edge Noise

Main noise mechanism of a clean wing is the turbulent boundary layer-trailing edge (TBL-TE) noise. Trailing edge noise originates primarily due to the scattering of some of the energy in the eddies directly into acoustic waves during the passage of a turbulent boundary layer over the trailing edge of wings or flaps (Figure 2.1). Turbulent pressure fluctuations in the wing boundary layer within an acoustic wavelength of the trailing edge are responsible for the noise generation. The spectrum of the trailing edge noise ranges from 100 Hz to over 10 kHz as shown in the experiments of Brooks et al.\textsuperscript{28,19}

The noise originating from the interaction of the turbulent flow with a sharp-edged body such as the trailing edge of a wing or a flap has been one of the main research areas of aeroacoustics for many years. Howe\textsuperscript{20} gives an extensive review of various trailing edge noise theories and lists them in different categories. He shows that, when appropriately interpreted, all theories given under different categories produce essentially identical trailing edge noise predictions for low Mach number flows. All theories on the trailing edge noise demonstrate that the far-field noise intensity varies approximately with the 5\textsuperscript{th} power of the free-stream velocity.\textsuperscript{20,11} It is also proportional to the trailing edge length along the span and a characteristic length scale of the turbulence.
Most of the theories used in predicting trailing edge noise are based on Lighthill’s Acoustic Analogy. Lighthill, in his theory of aerodynamic sound, modelled the problem of sound generation by turbulence in an exact analogy with sound radiated by a volume distribution of acoustic quadrupoles embedded in an ideal acoustic medium. In mathematical form, Lighthill’s analogy is the inhomogeneous wave equation written for the acoustic density fluctuations ($\rho'$):

$$\frac{\partial^2 \rho'}{\partial t^2} - a^2 \nabla^2 \rho' = \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}.$$  \hspace{1cm} (2.1)

Here, $a$ represents the speed of sound of the undisturbed fluid. The term $T_{ij}$ is Lighthill’s Turbulence Stress Tensor and can be approximated as the unsteady component of the Reynolds stress in low Mach number flows. In Lighthill’s analogy, the problem of calculating the aerodynamic sound is equivalent to solving Equation 2.1 for the radiation of sound into a stationary, ideal fluid produced by a distribution of quadrupole sources whose strength per unit volume is Lighthill’s stress tensor $T_{ij}$. An extensive and clear explanation of Lighthill’s Acoustic Analogy, including its derivation and implications, is given by Goldstein.

In Lighthill’s analogy, the turbulent fluctuations in free space (when there is no boundaries in the acoustic source regions) are inefficient radiators of noise in a low Mach number flow. In this case, the turbulent fluctuations are quadrupole type sources, therefore the radiated acoustic intensity in the far-field varies with $u_0^8$ where $u_0$ is a characteristic velocity scale. However, this character of the far-field noise intensity changes dramatically when the turbulent eddies pass in the vicinity of a sharp edge of a solid surface, and the radiation of the turbulent fluctuations are amplified significantly.

Ffowcs Williams and Hall were the first to investigate the problem of noise radiated from the turbulent flow past a semi-infinite plate of zero thickness at zero angle of attack. Their starting point was Lighthill’s Acoustic Analogy, and they sought a solution of Equation 2.1 when there is a rigid, vanishingly thin, half plane immersed in an otherwise unbounded fluid. They modelled a typical turbulent eddy as a quadrupole point source near the edge of the half plane. In their approach, the product $2kr_0$ was an important parameter where $k$ is the acoustic wave number $\omega_0/a$, $\omega_0$ is the radiant acoustic frequency, and $r_0$ is the distance from the quadrupole (or from the center of the eddy) to the edge of the half plane. They found that the sound output from the quadrupoles associated
with the eddies moving in a plane perpendicular to the edge which satisfy the inequality $2k r_0 \ll 1$ increases by a factor of $(k r_0)^{-3}$. Following this result, with the additional assumptions that the fluctuating component of eddy velocity and the acoustic frequency are linearly proportional to the characteristic velocity scale, Ffowcs Williams and Hall\textsuperscript{36} showed that the acoustic intensity increases by a factor of $u_0^{-3}$ relative to the case of quadrupole radiation in free space. This meant that the turbulent fluctuations in the vicinity of a sharp-edge radiate noise proportional to the fifth power of the characteristic velocity, which is the famous result of velocity scaling for the trailing edge noise. Ffowcs Williams and Hall also found that the the noise intensity has a directional dependence on $\sin^2(\theta/2)$ term in the far-field, where $\theta$ is the polar directivity angle measured relative to the downstream extension of the plate. For a more detailed analysis of the aerodynamic sound generation by turbulent flow in the vicinity of a scattering plane, the reader should refer to the original work by Ffowcs Williams and Hall\textsuperscript{36} or the book by Goldstein.\textsuperscript{38}

In their experimental study, Brooks and Hodgson\textsuperscript{19} measured the unsteady surface pressures at the trailing edge of a NACA 0012 airfoil model placed in an anechoic flow facility at low angles of attack. They mounted surface-pressure sensors near the airfoil trailing edge to obtain unsteady surface pressure data and measured the radiated noise with microphones at different angular positions and at different distances from the trailing edge. The results of their experimental study confirmed the velocity scaling and the directivity pattern of the trailing edge noise obtained by Ffowcs Williams and Hall.\textsuperscript{36} This confirmation showed the relevance of the results for the half-plane problem studied by Ffowcs Williams and Hall to more realistic problems involving airfoils and wings.\textsuperscript{10}

In a later study, Brooks \textit{et al.}\textsuperscript{28} presented an extensive experimental airfoil self-noise data set obtained with NACA 0012 airfoils having different chord lengths, at different angles of attack and different free-stream velocities. They investigated different noise source mechanisms including the turbulent boundary layer-trailing edge noise. Their data included the Sound Pressure Level (SPL) spectra of different noise sources. They also used this data set to develop a semi-empirical airfoil self-noise prediction method. Some of the test cases from their experimental study are used in the validation of the Noise Metric derived as part of the current study. The details of the Noise Metric validation are given in Chapter 4.

In recent years, Computational Aeroacoustics (CAA) methods have been used to simulate acoustic scattering from trailing edges. These methods couple time-accurate flow field
data obtained from RANS or Large Eddy Simulation solutions with acoustic equations to propagate the noise to the far-field. Singer et al.\textsuperscript{34} performed computational simulations of turbulence crossing an airfoil trailing edge, where the radiated noise has been computed using a time-accurate RANS solver coupled to Lighthill’s Acoustic Analogy\textsuperscript{31,32} in the form presented by Ffowcs Williams and Hawkings.\textsuperscript{30} Their results again confirmed the main results of the half-plane scattering problem studied by Ffowcs Williams and Hall.\textsuperscript{36} Other CAA studies on the simulation of trailing edge noise include the work by Ewert et al.\textsuperscript{39,40} and Lummer et al.\textsuperscript{41}

2.2 Derivation of the Noise Metric

The general outline of the Noise Metric derivation is given in Figure 2.2. As discussed in the previous section, both the experimental and computational aeroacoustics studies verify the relevance of modeling the TBL-TE noise created over the sharp trailing edges of airfoils and wings to the theoretical analysis of half-plane scattering problem studied by Ffowcs Williams and Hall. Therefore, in the derivation of the new Noise Metric, the results obtained from the Ffowcs Williams and Hall become the starting point. The originality of the current Noise Metric is in the modeling of the characteristic velocity and length scales in a way suitable for creating response surfaces used in design studies while capturing the important physics of the problem.

Following the approach by Goldstein,\textsuperscript{38} one can approximate the far-field noise intensity per unit volume of acoustic sources at the trailing edge of a wing as

$$ I \approx \frac{\rho_{\infty}}{2\pi^3 a_\infty^2 H^2} \omega_0 u_0^4 $$

where $\rho_{\infty}$ is the free-stream density, $a_\infty$ is the free-stream speed of sound, $\omega_0$ is the characteristic source frequency, $u_0$ is the characteristic velocity scale for turbulence, $H$ is the distance to the ground (receiver). This equation is a form of the Ffowcs Williams-Hall equation given by Goldstein.\textsuperscript{38} It is also similar to the form given in Howe\textsuperscript{20,37} and Crighton\textsuperscript{11} as also indicated by Lilley.\textsuperscript{12,42,10} Equation 2.2 gives the noise intensity at a point in the flyover plane where the polar angle ($\theta$) is 90°, and it is written for a trailing edge sweep angle ($\beta$) of zero. Therefore, it does not show the dependency of the noise intensity on the directivity and the trailing edge sweep angles. Following the approach
Figure 2.2: The general outline of the Noise Metric derivation

given in Howe,\textsuperscript{20} the trailing edge sweep angle dependency can be included by multiplying Equation 2.2 with the term $\cos^3 \beta$:

$$I \approx \frac{\rho_\infty}{2\pi^3 a_\infty^2 H^2} \omega_0 u_0^4 \cos^3 \beta \quad (2.3)$$

To write the noise intensity for any point in the far-field, a directivity term, $D(\theta, \psi)$ may be included in the above equation to give:

$$I \approx \frac{\rho_\infty}{2\pi^3 a_\infty^2} \omega_0 u_0^4 \cos^3 \beta \frac{D(\theta, \psi)}{H^2} \quad (2.4)$$

Here, the directivity term is in the form given by Ffowcs Williams and Hall:\textsuperscript{36}

$$D(\theta, \psi) = 2\sin^2\left(\frac{\theta}{2}\right)\sin \psi \quad (2.5)$$

where $\theta$ is the polar directivity angle and $\psi$ is the azimuthal directivity angle. (Figure 2.3).

The Doppler factors due to convection of acoustic sources are not included in Equation 2.4, since the focus of the current study is on flows with low Mach numbers which are between 0.2 and 0.3 for typical aircraft at approach before landing. As indicated by Lilley,\textsuperscript{12,10} the equivalent noise sources in the wing boundary layer are in motion relative
to the wing, therefore they appear to be moving very slowly to an observer on the ground. The relative velocity between the sources and the observer determines the magnitude of the Doppler factors. Since the relative velocity is small for the cases considered in this study, the Doppler factors may be omitted.

Using the Strouhal relation for turbulent flow,\textsuperscript{12}

\[
\frac{w_0l_0}{u_0} \approx \text{const}
\]  \hspace{1cm} (2.6)

one can re-write Equation 2.4 with the characteristic length scale for turbulence \( l_0 \):

\[
I \approx \frac{\rho_{\infty}u_0^5}{2\pi^3a_{\infty}^2}w_0^{-1}\cos^3\beta \frac{D(\theta, \psi)}{H^2}
\]  \hspace{1cm} (2.7)

Since it is desired to design a wing for minimum noise, one should consider the spanwise variation of the characteristic velocity, characteristic length scale, the trailing edge sweep, and the directivity angles (i.e., \( u_0 = u_0(y) \), \( l_0 = l_0(y) \), \( \beta = \beta(y) \), \( \theta = \theta(y) \), and \( \psi = \psi(y) \)). The importance of retaining the spanwise variation of the characteristic velocity and length scale can be seen in the three-dimensional parametric studies given in Chapter 6, since the changes in these variables are significant along the span at higher lift coefficients.
Assuming a correlation volume per unit span at the trailing edge as

\[ dV = l_0^2 dy \]  

Equation 2.7 can be written for the correlation volume given above and integrated over the span \( b \) to obtain

\[ I_{NM} = \frac{\rho_\infty}{2\pi^3 a_\infty^2} \int_0^b u_0^5 l_0 \cos^3 \beta \frac{D(\theta, \psi)}{H^2} dy \]  

where \( I_{NM} \) is a noise intensity indicator which can be evaluated on the upper or the lower surface of the wing. Note that \( I_{NM} \) is not the absolute value of noise intensity, however it is expected to be an accurate indicator as a relative noise measure. The noise intensity indicator \( I_{NM} \) is scaled with the reference noise intensity of \( 10^{-12} \, W/m^2 \) (i.e, the minimum sound intensity level that human ear can detect, which is a common practice in acoustics). Finally, the proposed Noise Metric (NM) for the trailing edge noise (in dB) can be written as:

\[ NM = 120 + 10 \log(I_{NM}) \]  

To obtain the total Noise Metric for a wing, the Noise Metric values are calculated for the upper \( (NM_{upper}) \) and the lower surfaces \( (NM_{lower}) \), and added as:

\[ NM = 10 \log \left( 10^{\frac{NM_{upper}}{10}} + 10^{\frac{NM_{lower}}{10}} \right) \]  

2.3 Modeling of \( u_0 \) and \( l_0 \)

In the new Noise Metric, the characteristic turbulent velocity at a spanwise location of the wing trailing edge can be chosen as the maximum value of the turbulent kinetic energy \( (TKE) \) profile at that particular spanwise station:

\[ u_0(y) = \text{Max} \left[ \sqrt{TKE(z_n)} \right] \]  

Here, \( z_n \) is the direction normal to the wing surface. Others have proposed the same choice for the characteristic velocity in their noise models.\(^{10}\) It is proposed here, that the characteristic turbulence length scale for each spanwise station can be well represented
by

\[ l_0(y) = \frac{\text{Max}\left[\sqrt{TKE(z_n)}\right]}{\omega} \]  \hspace{1cm} (2.13)

In Equation 2.13, \( \omega \) is the turbulence frequency (dissipation rate per unit kinetic energy) observed at the maximum \( TKE \) location. This choice of a length scale is directly related to the turbulent characteristics of the flow and is indicative of the size of the turbulent eddies that produce the noise. It can be viewed as more soundly based than other suggestions in the literature like the boundary layer thickness or the displacement thickness. Those lengths are related to the mean flow and reflect little about the turbulence structure. The turbulent kinetic energy (\( TKE \)) and the turbulence frequency (\( \omega \)) are obtained from the solutions of the \( TKE-\omega \) (\( k-\omega \)) turbulence model equations used in the Reynolds Averaged Navier-Stokes calculations. The details of the CFD simulations are given in Chapter 3. Appendix A gives an extensive description of the method used in extracting \( u_0 \) and \( l_0 \) from the results of the CFD simulations for two- and three-dimensional problems.

### 2.4 Lilley’s Clean Aircraft Noise Formulation

In his 2001 paper, Lilley gives the following expression to approximate the far-field noise intensity radiated from a clean aircraft:

\[ I = K \left( \frac{W V_\infty M^2_\infty}{C_L H^2} \right) \]  \hspace{1cm} (2.14)

Here, \( V_\infty \) (\( m/s \)) is the free-stream velocity, \( M_\infty \) is the free-stream Mach number, \( W \) (\( Newtons \)) is the weight of the aircraft, \( C_L \) is the overall lift coefficient of the aircraft, and \( H \) (\( m \)) is distance to the observer (altitude). \( K \) is a constant, which is equal to \( 5.6 \times 10^{-7} \). This equation assumes that the noise of the clean aircraft originates only from the trailing edge of the wing. Lilley\textsuperscript{12} starts his derivation from Equation 2.2, which is the far-field noise intensity per unit volume of acoustic sources (or turbulence) at the trailing edge. This expression is a form of the Ffowcs Williams and Hall equation given by Goldstein.\textsuperscript{38} It should be noted that the derivation of the proposed Noise Metric here also starts from this equation. However, Lilley continues the derivation by considering the flyover
case with a polar directivity angle ($\theta$) of 90° which makes the directivity term $D(\theta, \psi)$ of Equation 2.5 equal to unity. He also ignores the $\cos^3 \beta$ term given by Howe\textsuperscript{20} since the contribution of this term is small for most conventional wings. However, Lilley\textsuperscript{42} also states that the radiated noise from scattering may be reduced to a smaller value for wings with highly swept trailing edges. After using the Strouhal relation given in Equation 2.6, Lilley\textsuperscript{12} re-writes Equation 2.2 in terms of the characteristic length scale ($l_0$) and the velocity scale ($u_0$) for turbulence. He uses the displacement thickness ($\delta^*$) at the trailing edge for the length scale and the square-root of the turbulent kinetic energy for the characteristic velocity. Lilley then integrates this form of Equation 2.2 written in terms of the length and the velocity scales over the wing span. Using the equation

$$W = \frac{1}{2}\rho_\infty V_\infty^2 SC_L$$

written for an aircraft of weight $W$, flying straight and level before the approach, he includes $W, C_L$ in his final expression given by Equation 2.14.

It should be noted that Lilley\textsuperscript{12} assumes constant values of the characteristic velocity and the length scale along the span in his formula (Equation 2.14). In fact, these values are used to obtain the coefficient $K$. In the three-dimensional parametric Noise Metric studies presented in Chapter 6, significant variations of the velocity and length scales, especially at high lift coefficients, can be seen. Furthermore, this form of Lilley’s formulation does not take into account the change of the velocity and the length scale with the lift coefficient $C_L$. It will be shown that, for $C_L > 0.5$, the changes in the turbulent kinetic energy and the length scale start to become significant so these parameters can no longer be assumed to be constant. As shown in the parametric studies, the Noise Metric derived as part of the current work captures the change in the velocity and the length scale as the lift coefficient increases.

In a later study, Lockard and Lilley\textsuperscript{10} modify the formula given by Equation 2.14 to include $C_L$ effect on the characteristic velocity and the length scale. In their approach, Lockard and Lilley\textsuperscript{10} also use the location of the maximum turbulent kinetic energy (in our convention, $(z_n)_{\text{max}}$) as the characteristic length scale, since the displacement thickness can no longer be assumed to be a reasonable value for this purpose. They use a CFD database of RANS simulations performed on NACA 4412 airfoil at incidences
changing from zero-lift to stall to obtain the following functional relation:

\[
\left[ \left( \frac{u_0}{V_\infty} \right)^5 \frac{\delta}{(z_n)_{max}} \right]_{TE} \propto \left( 1 + \frac{1}{4} C_L^2 \right)^4 \tag{2.16}
\]

Here, \( \delta \) is the boundary layer thickness. The left-hand-side of the equation is evaluated at the trailing edge (TE) of the airfoil. By using this result, they obtain a modified version of Equation 2.14:

\[
I = K \frac{V_\infty M_\infty^2 W}{C_L H^2} \left( 1 + \frac{1}{4} C_L^2 \right)^4 \tag{2.17}
\]

Lockard and Lilley\(^{10}\) use this modified form to approximate the far-field noise intensity from a clean wing at high lift. However, this equation still does not take into account the spanwise variation of the velocity and length scales, which become important at high lift coefficients for three-dimensional cases.

### 2.5 ANOPP Clean Wing Noise Model

NASA Langley’s Aircraft Noise Prediction Program\(^{27}\) (ANOPP) uses Fink’s Method\(^{26}\) in its airframe noise prediction module. In ANOPP, airframe noise sources include the clean wing, tail, landing gear, flaps and leading edge slats. This section gives a brief description of the clean wing noise prediction. The reader should refer to the report by Fink\(^{26}\) for a detailed derivation and explanation of the method.

Fink’s prediction\(^{27,26}\) for broadband noise from a clean wing includes a semi-empirical function to calculate the mean square acoustic pressure \( \overline{p^2(f, \theta, \psi)} \) as a function of frequency \( f \), polar directivity angle \( \theta \), and azimuthal directivity angle \( \psi \) for a given flight condition:

\[
\overline{p^2(f, \theta, \psi)} = \frac{K \rho_\infty V_\infty^5 \delta_{wb}}{4\pi H^2 \alpha_\infty} \left( \left[ \text{St}(f, \theta) \right] \frac{D(\theta, \psi)}{(1 + M_\infty \cos \theta)^4} \right) \tag{2.18}
\]

Note that above equation is written to predict the total noise originating from the upper and the lower surface of a clean wing. Here, the directivity function is given by

\[
D(\theta, \psi) = 4 \sin^2 \left( \frac{\theta}{2} \right) \sin^2 \psi \tag{2.19}
\]
The definitions of the directivity angles are shown in Figure 2.3. In equation 2.18, $K$ is an empirical non-dimensional constant, which is equal to $7.075 \times 10^{-6}$ for an aerodynamically clean wing. This constant includes the turbulence intensity within the boundary layer which was assumed to be independent of Reynolds number for conditions that are typical of aircraft wings. In the same equation, $V_\infty$ is the free-stream velocity, $M_\infty$ is the free-stream Mach number, $H$ is the distance to the observer, and $a_\infty$ is the free-stream speed of sound. The characteristic length scale for turbulence is taken as the boundary layer thickness at the wing trailing edge and is computed from a standard flat-plate turbulent boundary layer thickness approximation model:

$$\delta_w = 0.37 \frac{S_{ref}}{b} \left[ \frac{S_{ref} V_\infty}{\nu b} \right]^{-0.2}$$  \hspace{1cm} (2.20)

The spectrum function $F[St(f, \theta)]$ is determined empirically and given by

$$F[St(f, \theta)] = 0.613 \left( 10St(f, \theta) \right)^4 \left[ \left( 10St(f, \theta) \right)^{1.5} + 0.5 \right]^{-4}$$  \hspace{1cm} (2.21)

for a rectangular wing. Fink also gives a modified version of this function for delta wings. Here Strouhal number is a function of the frequency (Hz) and the polar directivity angle $\theta$ for fixed flow conditions and is defined as

$$St(f, \theta) = \frac{f \delta_w V_\infty}{1 + M_\infty \cos \theta}$$  \hspace{1cm} (2.22)

Using the definitions above, the overall mean square acoustic pressure at a given location can be obtained by integrating the contributions from all the frequencies.

$$\overline{p^2} = \frac{K \rho_\infty^2 V_\infty^5 \delta_w b}{2\pi H^2 a_\infty} \int_0^\infty F(f) df$$  \hspace{1cm} (2.23)

Note that Equation 2.23 is written for a location in the flyover plane ($\psi = 90^\circ$) with a polar directivity angle ($\theta$) of $90^\circ$. Here the spectrum function $\tilde{F}(f)$ is written using the definition of the Strouhal number (Equation 2.22) in Equation 2.21:

$$\tilde{F}(f) = 0.613 \left( \frac{10f \delta_w}{V_\infty} \right)^4 \left[ \left( \frac{10f \delta_w}{V_\infty} \right)^{1.5} + 0.5 \right]^{-4}$$  \hspace{1cm} (2.24)

After obtaining the overall mean square acoustic pressure, the far-field noise intensity ($I$)
at the same location is calculated by

\[ I = \frac{p^2}{\rho \omega a} \]  

(2.25)

Finally the Overall Sound Pressure Level (in dB) is obtained by scaling the noise intensity with the reference noise intensity value of \(10^{-12} \text{ Watts/m}^2\)

\[ OASPL = 120 + 10\log(I) \]  

(2.26)

Since Equation 2.20 is used to approximate the boundary layer thickness for a flat-plate, it does not take into account the change of the boundary layer thickness with the lift coefficient. In ANOPP clean wing noise module, the turbulence intensity is also assumed to be independent of the change in Reynolds number and the lift coefficient. Since the characteristic length and velocity scales used in this model do not vary with the lift coefficient, the clean wing noise prediction is also independent of the change of the lift coefficient. The ANOPP clean wing noise model is derived mainly to predict the noise at lower lift coefficients (between \(C_L = 0.2\) and 0.6) as indicated by Fink.\(^{26}\) The effect of the lift coefficient on the clean wing noise at lower \(C_L\) values is small, however the increase in noise can be significant at higher lift coefficients as will be shown in two- and three-dimensional studies given in Chapters 5 and 6.

### 2.6 Unique Features of the Proposed Noise Metric

The new Noise Metric is developed here in a way that could be used in the optimization problems involving aerodynamic noise from a clean wing. The Noise Metric is not the absolute value of the noise intensity, however it has been shown to be an accurate noise indicator by the validation studies given in Chapter 4. The unique features of this new noise measure can be summarized as follows:

- The current Noise Metric can be applied to any clean wing geometry, the rotor blades of helicopters, or the blades used in the wind turbines. Many of the practical trailing edge noise prediction methods used today are based on semi-empirical relations. In these methods, the characteristic length and velocity scales are usually
determined from curve fits obtained from experiments or flight measurements. The empirical nature of these methods may limit the accuracy level of their predictions in cases where the problem variables (flow conditions, geometries, etc.) are different than the range of parameters used for building the empirical database. One of the benefits of the new Noise Metric approach is to be able model characteristic velocity and length scale by using CFD RANS solutions to achieve better noise prediction for different flow conditions and geometries.

- All trailing edge noise theories, as reviewed by Howe\textsuperscript{20} and Crighton,\textsuperscript{11} show that the far-field noise intensity is proportional to the characteristic length scale for turbulence. This fact emphasizes the importance of obtaining the length scale as accurately as possible for better noise prediction. The length scale used in the Noise Metric is a better indicator of the turbulence structure at the trailing edge of an airfoil or wing than the other quantities suggested in the literature such as the boundary layer or the displacement thickness. Those lengths are related to the mean flow, and reflect very little about the turbulence structure, while the current suggestion is directly related to flow turbulence.

- The proposed Noise Metric takes into account the spanwise variation of the turbulent kinetic energy (or the characteristic velocity) and the length scale. The changes in velocity and length scale along the span become significant especially at high lift coefficients as shown by the three-dimensional parametric studies (Chapter 6). The models based on two-dimensional assumptions fail to capture the three-dimensional effects, which may affect the accuracy of the noise prediction.

- The new Noise Metric was developed so that it can capture the effect of different design variables on the trailing edge noise. Besides including some parameters explicitly in the model (directivity angles $\theta$ and $\psi$, distance to the observer $H$, and trailing edge sweep $\beta$), the current Noise Metric is also capable of capturing the effect of the other design variables such as the lift coefficient $C_L$ and the wing geometry (thickness ratio $t/c$, airfoil shape, twist, trailing edge sweep, etc), since the changes in these variables affect the characteristic velocity and the length scale which are obtained from the RANS simulations. Most empirical noise prediction methods ignore the effects of such parameters on the velocity and the length scale.
Chapter 3

CFD Simulations

3.1 Governing Equations

The basic flow equations solved in the noise metric studies are the steady Reynolds Averaged Navier-Stokes (RANS) equations. The flow of a viscous, heat conducting, compressible, single-species, non-reacting continuous fluid can be described by the Navier-Stokes equations which consist of the mass continuity equation (conservation of mass), Newton’s Second Law of motion (conservation of momentum), and the first law of thermodynamics (conservation of energy). For turbulent flows, these equations can be re-written by using Reynolds averaging (shown with a over bar) for the density ($\rho$) and pressure ($p$), and mass-weighted averaging (shown with a tilde) for the velocity components ($u_i$) and temperature ($T$). In general if $a$ is a flow quantity, it can be written as $a = \bar{a} + a'$ with Reynolds Decomposition where $a'$ is the fluctuating component, and $a = \tilde{a} + a''$ with Favre Averaging where $a''$ represents the fluctuating part. Note that for incompressible flow where the density is constant, Reynolds decomposition and Favre Averaging are essentially the same. Using these definitions, the most general form of RANS equations can be given as:

Conservation of Mass:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_i} = 0$$  \hspace{1cm} (3.1)
Conservation of Momentum:
\[
\frac{\partial \tilde{p} \tilde{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\tilde{p} \tilde{u}_i \tilde{u}_j + \tilde{p} \delta_{ij}) = \frac{\partial}{\partial x_j} (\tilde{\tau}_{ij} + \tilde{\tau}_{ij}^{''}) - \frac{\partial}{\partial x_j} \left( \tilde{p} \tilde{u}^{''}_i \tilde{u}^{''}_j \right) \tag{3.2}
\]

Conservation of Energy:
\[
\frac{\partial \tilde{p} \tilde{e}_0}{\partial t} + \frac{\partial}{\partial x_i} (\tilde{p} \tilde{e}_0 \tilde{u}_i + \tilde{p} \tilde{u}_i + \tilde{p} \tilde{e}_0^{''} u_i^{''}) = \frac{\partial}{\partial x_i} (\tilde{\tau}_{ij} u_j) - \frac{\partial q_i}{\partial x_i} \tag{3.3}
\]

where \(e_0\) is the total specific energy and defined as
\[
\tilde{e}_0 = C_v \tilde{T} + \frac{1}{2} \tilde{u}_i \tilde{u}_i + \frac{1}{2} \tilde{u}^{''}_i \tilde{u}^{''}_i \tag{3.4}
\]
Here, \(C_v\) is the specific heat at constant volume. The perfect-gas law is used to close the system.
\[
\tilde{p} = \tilde{p} \tilde{R} \tilde{T} \tag{3.5}
\]
where \(R\) is the gas constant. In equations 3.2 and 3.3, \(\tau_{ij}\) is viscous shear stress tensor and \(q_i\) is the heat flux vector component. The term \(-\tilde{p} \tilde{u}^{''}_i \tilde{u}^{''}_j\) is called the Reynolds stress tensor. In CFD simulations, this term is approximated with turbulence models. All eddy viscosity turbulence models assume the Boussinesq approximation for the Reynolds stress tensor:
\[
-\tilde{p} \tilde{u}^{''}_i \tilde{u}^{''}_j = \mu_t \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \tilde{u}_k}{\partial x_k} \right) - \frac{2}{3} \delta_{ij} \tilde{p} \tilde{k} \tag{3.6}
\]
where \(\mu_t\) is the eddy viscosity and \(k\) (TKE) is the turbulent kinetic energy given by
\[
k = \frac{1}{2} \frac{\tilde{p} \tilde{u}^{''}_i \tilde{u}^{''}_j}{\tilde{p}} \tag{3.7}
\]

### 3.2 Numerical Solver

The numerical solver used in this study was the General Aerodynamics Simulation Program\textsuperscript{43,44} (GASP). The numerical solutions of all validation and parametric Noise Metric cases were obtained using GASP, which is a three-dimensional, structured, multi-block, finite volume, Reynolds-Averaged Navier-Stokes (RANS) code. GASP is capable of solving the integral form of the time-dependent RANS equations for flow problems ranging from
subsonic to hypersonic speeds in three-dimensions as well as the subset of RANS equations such as the Thin layer Navier-Stokes (TLNS), Parabolized Navier-Stokes (PNS), or Euler Equations. The code has different flux splitting options, turbulence models, discretization, and time integration schemes. GASP can be run in most parallel computers, as well as clusters of workstations.

3.3 Physical Modeling

In the CFD simulations, inviscid fluxes were calculated by an upwind-biased third-order spatially accurate Roe flux scheme in all dimensions. Asymptotic convergence to a steady state solution was obtained for each case using a fully implicit three-dimensional Symmetric Block Gauss Seidel solver.

Each case was run at three grid levels: coarse, medium, and fine. Medium and coarse grid levels were obtained from the fine grid by reducing the number of grid points by a factor of two at each direction. At each grid level, except the coarsest one, the initial solution estimates were obtained by interpolating the primitive variable values of the previous grid solution to the new cell locations. This method, known as grid sequencing, was used to reduce the number of iterations required to converge to a steady state solution at finer mesh levels. All the results presented in this study were obtained with the finest mesh level.

For the airfoil and the wing problems, the Riemann subsonic inflow/outflow boundary condition was used at the far-field boundaries. No-slip adiabatic wall boundary condition was applied to the airfoil and wing surfaces. For the wing cases, the symmetry condition was set for the root section plane.

The iterative convergence of each solution was examined by monitoring the overall residual, which is the sum (over all the cells in the computational domain) of the $L^2$ norm of all the governing equations solved in each cell. In addition to this overall residual information, some of the output quantities such as the lift coefficient and the $TKE$ values at selected cell centers were also monitored. Figure 3.1 shows an example of the iteration history of the lift coefficient from a wing case at the fine grid level performed in three-dimensional parametric studies.
In the CFD simulations, all viscous terms were retained in the physical model (no thin-layer approximation was made). The viscous terms were discretized using a standard second-order accurate central differencing scheme. All the runs were made with the assumption of fully-turbulent flow. Menter’s $k$-$\omega$ SST turbulence model\textsuperscript{46} was used in all the calculations.

### 3.3.1 Menter’s $k$-$\omega$ SST Turbulence Model

Menter’s $k$-$\omega$ shear stress transport (SST) model is a two-equation eddy viscosity turbulence model. Here $\omega$ is the turbulence frequency (or the specific dissipation rate of turbulent kinetic energy) used in the calculation of the length scale ($l_0$) in the Noise Metric (See Section 2.3). This model has been shown\textsuperscript{47} to give better overall accuracy in different types of flows, especially in adverse pressure gradient and separated flows, compared to the other two-equation turbulence models. In his SST model, Menter\textsuperscript{46} combined the best features of traditional $k$-$\omega$ and $k$-$\epsilon$ models. He did this by first transforming the $k$-$\epsilon$ model to $k$-$\omega$ form. He then modified the constants of the resulting model to follow the Wilcox $k$-$\omega$ model in the inner or wall region of the boundary layer and
left intact the $k$-$\epsilon$ constants in the outer regions and in free shear layers. By doing so, he retained the near wall advantages of the $k$-$\omega$ model while eliminating its free-stream sensitivity at the boundary layer edge. A detailed description of the model can be found in Menter.⁴⁶

3.4 Computational Grids

The computational grids used in the CFD simulations were obtained using the grid generation program Gridgen.⁴⁸ The two-dimensional structured grids were created around airfoils that were used in the validation and two-dimensional parametric Noise Metric studies. The three-dimensional structured grids were prepared around conventional transport wings that were used in the three-dimensional parametric Noise Metric studies.

3.4.1 Two-Dimensional Grids

The two-dimensional grids were created for two subsonic airfoils (NACA 0009 and NACA 0012) and two supercritical airfoils (SC(2)-0710 and SC(2)-0714). All airfoils were used in the two-dimensional parametric Noise Metric studies. NACA 0012 airfoil was used also in the Noise Metric validation cases (Chapter 4). The geometric details of the airfoils can be found in Chapter 5. All two-dimensional grids were created with the same C-topology shown in Figure 3.2. The two-dimensional grids have 389 points in the streamwise ($I$) direction and 65 points in the direction normal to airfoil surface ($J$). The wake region of the airfoil contains 45 points. The grid points were clustered at the trailing edge and the leading edge of the airfoil to give a minimum streamwise spacing of $0.001 \times$ chord length. In turbulent boundary layers, a $z_n^+$ value between 7 and 10 is considered as the edge of the laminar sublayer. Here

$$ z_n^+ = \frac{z_n u_*}{\nu} \quad (3.8) $$

where $z_n$ is the normal distance from the wall, $u_*$ = $\sqrt{\tau_w/\rho}$ is the friction velocity, $\tau_w$ is the wall shear stress, $\rho$ is the density, and $\nu$ is the kinematic viscosity. General CFD practice has been to have several mesh points in the laminar sublayer with the first mesh point at $z_n^+ = O(1)$. In all cases performed in this study, the $z_n^+$ value of the first grid
Figure 3.2: The C-grid topology used in the two-dimensional airfoil cases.

point from the wall at the upper surface trailing-edge of the airfoil was less than 0.5. Figure 3.3 shows an example $z_n^+$ distribution on the upper surface of the SC(2)-0714 airfoil at a lift coefficient of $C_l = 1.665$. As can be seen from this figure, all $z_n^+$ values on the upper surface of the airfoil are less than 1.0. Figure 3.4 shows a portion of the grid around the NACA 0012 airfoil and gives a close-up view of the trailing edge region. A similar view of the SC(2)-0714 airfoil is shown in Figure 3.5.

3.4.2 Three-Dimensional Grids

The three-dimensional grids were created for the two versions of the same transport wing (EET Wing\textsuperscript{49}) used in the parametric Noise Metric studies. The original wing had a baseline twist distribution. The baseline twist distribution was changed to obtain a modified wing which was used to investigate the effect of the twist on the trailing edge noise. All the other geometric wing parameters were kept the same. The details of
the EET wing geometry are presented in Chapter 6. The computational grids of both wings had the same dimensions and spacing parameters. Each grid was created using a C-O topology consisting of four blocks with a total number of 884,736 cells (Figures 3.6 and 3.7). The grids had 64 cells in the normal direction to the wing surface, 216 points in the streamwise direction, and 64 points in the spanwise direction. Figure 3.8 shows the planform view of the original wing grid. As can be seen from this figure, the grid points along the span were clustered at the wing break and wing tip locations. In the streamwise direction, the clustering was done on the leading and the trailing edge regions of the wing.
Figure 3.4: The grid around the NACA 0012 airfoil ($c = 0.3048 \text{ m}$) used in the CFD simulation of the validation cases. A close-up view of the trailing edge region is given on the right. Every other grid line in the streamwise direction is shown.

Figure 3.5: The grid around the SC(2)-0714 airfoil ($c = 9.54 \text{ m}$) used in the two-dimensional parametric Noise Metric studies. A close-up view of the trailing edge region is given on the right. Every other grid line in the streamwise direction is shown.
Figure 3.6: A view of the original wing and the C grid around the root section.

Figure 3.7: A view of the grid in the wing tip region.
Figure 3.8: The planform view of the original wing grid used in the three-dimensional Noise Metric studies.
The validation studies were performed to test the accuracy of the Noise Metric predictions at different flow conditions. The Noise Metric results were compared to the experimental test cases selected from the airfoil noise database presented by Brooks et al.\textsuperscript{28} For the same validation cases, noise predictions obtained using the clean wing noise model of ANOPP\textsuperscript{27} were also compared to the experimental data. In this chapter, the experimental test cases and the corresponding CFD simulations for obtaining the Noise Metric are described in detail. Following the descriptions, the validation strategy and the results of the validation studies are presented. The validation studies show that the Noise Metric is an accurate relative noise indicator, which is capable of capturing the variations in the trailing edge noise when different parameters and flow conditions are changed.

4.1 Description of the Experimental Data

Among the experimental trailing edge noise studies in the literature, the work by Brooks et al.\textsuperscript{28} gives the most extensive experimental database. To create this data set, Brooks et al.\textsuperscript{28} conducted experiments at different speeds, angles of attack, and chord lengths using NACA 0012 airfoils and measured the 1/3-octave Sound Pressure Level (SPL) spectra of the noise generated by the airfoils. They also used this database to develop a semi-empirical airfoil noise prediction method.

The NACA 0012 models used in the experiments were tested in the low-turbulence poten-
tial core of a free jet located in an anechoic chamber. The jet was provided by a vertically mounted nozzle with a rectangular exit. The details of the experimental facility can be found in Brooks et al.\textsuperscript{28,50,51} Six two-dimensional NACA 0012 models with different chord lengths (2.54, 5.08, 10.16, 15.24, 22.86, and 30.48 cm) were used in the experiments. The span lengths of all the models were 45.72 cm. The models were made with very sharp trailing edges, less than 0.05 mm thick. Brooks et al.\textsuperscript{28} also used two-dimensional models with thick trailing edges to study the vortex-shedding noise associated with bluntness of the trailing edge. However, since the main interest of the current study is to evaluate the accuracy of a trailing edge Noise Metric, only the experiments made with the models that have sharp-edge trailing edges have been considered for the validation work here.

The models were tested at four different free-stream velocities: 31.7, 39.6, 55.5, and 71.3 m/s, which correspond to the free-stream Mach numbers of 0.092, 0.116, 0.162, and 0.208. Based on the chord length of 0.3048 m and a free-stream velocity of 71.3 m/s, $1.497 \times 10^6$ was the highest Reynolds Number achieved in the experiments. The tests were performed at different angles of attack starting from zero lift. The testing of the airfoil models in a finite-size open wind tunnel causes flow curvature and downwash deflection of the incident flow that does not occur in free-air.\textsuperscript{28} This reduces the angle of attack, especially for larger models. Brooks et al. presented their experimental results at effective angles of attack ($\alpha^*$, the angle that would give the same lift in free-air) which were obtained by using a correction method.\textsuperscript{51} The CFD simulations used the corrected angles of attack to obtain the Noise Metric for selected cases.

The experiments included acoustic and aerodynamic measurements. The acoustic measurements were performed with eight, free-field-response microphones mounted in the plane perpendicular to the two-dimensional model mid-span. These microphones were used to determine the spectra for noise from airfoils encountering smooth air flow. Each microphone was 1.22 m away from the mid-span trailing edge. The results of the noise measurements were presented as $1/3$-octave Sound Pressure Level (SPL) spectra.
The aerodynamic measurements were performed with hot-wire probes in the boundary layer/near-wake region of the sharp trailing edge of the two-dimensional airfoil models. The probes were traversed perpendicular to the model chord lines at the trailing edge to obtain the velocity profiles. The boundary layer quantities such as the boundary layer thickness, displacement thickness, and the momentum thickness were calculated by using the measured velocity profiles. The reader should refer to the NASA Reference Publication by Brooks et al.\textsuperscript{28} for more details of the acoustic and aerodynamic measurements including the instrumentation, data acquisition and reduction.

### 4.2 Outline of the Selected Experimental Test Cases

Noise Metric validation was performed with seven test cases selected from the experimental database of Brooks et al.\textsuperscript{28} The effective angle of attack ($\alpha^*$), airfoil chord length ($c$), free-stream velocity ($V_\infty$), free-stream Mach number ($M_\infty$), and the Reynolds number based on the chord length ($Re_c$) for each case are listed in Table 4.1. The first case was used as the reference case in the calculation of the relative Overall Sound Pressure Level (experimental) and the Noise Metric (computational) values. The following points were effective in the selection process:

- The selected test cases enable the evaluation of the accuracy of the Noise Metric at different flow conditions, since they cover a wide range of speeds at different angles of attack. The Reynolds Number changes depending on the speed and the chord length used.

- The main noise mechanism of all these test cases is the scattering of turbulent pressure fluctuations over the trailing edge.

- The selection of cases with high angles of attack was avoided in order to limit the uncertainty coming from the angle of attack correction. Also, at very high angles of attack, Brooks et al.\textsuperscript{28} show massive flow separation starting from regions closer to the leading edge of airfoil models. In such cases, the uncertainties associated with the CFD simulations increase as shown in Appendix B.

In their report, Brooks et al.\textsuperscript{28} plotted the $1/3$-octave band $SPL$ spectra for each airfoil case. However, the data were not tabulated. In order to reduce the uncertainty coming
Table 4.1: Experimental NACA 0012 airfoil test cases used in the Noise Metric Validation.

<table>
<thead>
<tr>
<th>Case (i)</th>
<th>(\alpha^*) (deg.)</th>
<th>Chord length (m)</th>
<th>(V_\infty)</th>
<th>Mach</th>
<th>(Re_c \times 10^{-6})</th>
</tr>
</thead>
<tbody>
<tr>
<td>1*</td>
<td>0.00</td>
<td>0.3048</td>
<td>71.3</td>
<td>0.208</td>
<td>1.497</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.3048</td>
<td>31.7</td>
<td>0.092</td>
<td>0.665</td>
</tr>
<tr>
<td>3</td>
<td>2.00</td>
<td>0.2286</td>
<td>31.7</td>
<td>0.092</td>
<td>0.499</td>
</tr>
<tr>
<td>4</td>
<td>1.50</td>
<td>0.3048</td>
<td>39.6</td>
<td>0.116</td>
<td>0.831</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>0.3048</td>
<td>55.5</td>
<td>0.162</td>
<td>1.164</td>
</tr>
<tr>
<td>6</td>
<td>2.00</td>
<td>0.2286</td>
<td>71.3</td>
<td>0.208</td>
<td>1.122</td>
</tr>
<tr>
<td>7</td>
<td>1.50</td>
<td>0.3048</td>
<td>71.3</td>
<td>0.208</td>
<td>1.497</td>
</tr>
</tbody>
</table>

from the reading of the graphs, the semi-empirical noise prediction method based on the experimental data of the same airfoil studies was used to obtain the \(SPL\) spectra. For the selected test cases, the difference between the experimental data and the values obtained with the semi-empirical prediction method was very small. The following section gives a brief description of this semi-empirical airfoil noise prediction method.

### 4.2.1 Semi-Empirical Airfoil Noise Prediction

Brooks et al.\textsuperscript{28} used the measured noise spectra and boundary layer quantities obtained from their NACA 0012 experiments to develop a semi-empirical airfoil noise prediction method. Measured boundary layer data were used to construct curve fits for predicting the boundary layer thickness, displacement thickness, and the momentum thickness at different Reynolds Numbers and angles of attack for tripped and untripped cases. The boundary layer quantities obtained from these curve fits were then used in the airfoil noise prediction equations.

Brooks et al.\textsuperscript{28} identified different noise mechanisms due to specific boundary layer phenomena. Among these, the mechanism relevant to the current study is described as the *boundary layer turbulence passing over the trailing edge*. The noise created by this mechanism is the turbulent boundary layer-trailing edge noise as discussed in Chapter 2. For each noise mechanism, Brooks et al.\textsuperscript{28} developed a noise prediction procedure. They developed the following equation to predict the 1/3-octave SPL spectrum for the trailing
edge noise originating from the pressure-side of the airfoil:

\[
SPL_p = 10 \log \left( \frac{\delta_p^* M_\infty^5 b \bar{D}_h}{H^2} \right) + A \left( \frac{St_p}{St_1} \right) + (K_1 - 3) + \Delta K_1. \tag{4.1}
\]

A similar equation was written for the suction-side contribution:

\[
SPL_s = 10 \log \left( \frac{\delta_s^* M_\infty^5 b \bar{D}_h}{H^2} \right) + A \left( \frac{St_s}{St_1} \right) + (K_1 - 3). \tag{4.2}
\]

For non-zero angles of attack \((\alpha^* > 0)\), in addition to Equations 4.1 and 4.2, the following equation was used to improve the prediction of the total trailing edge noise.

\[
SPL_\alpha = 10 \log \left( \frac{\delta_s^* M_\infty^5 b \bar{D}_h}{H^2} \right) + B \left( \frac{St_s}{St_2} \right) + K_2 \tag{4.3}
\]

The total 1/3-octave SPL spectrum of the trailing edge noise was obtained by the logarithmic summation of \(SPL_p\), \(SPL_s\), and \(SPL_\alpha\):

\[
SPL_{TOT} = 10 \log \left( 10^{SPL_p/10} + 10^{SPL_s/10} + 10^{SPL_\alpha/10} \right) \tag{4.4}
\]

Note that \(SPL_p\), \(SPL_s\), \(SPL_\alpha\), and \(SPL_{TOT}\) are all functions of the 1/3-octave band frequencies and give the sound pressure level at each frequency band in Decibels (dB). In equations 4.1, 4.2 and 4.3, \(\delta_p^*\) is the pressure-side displacement thickness and \(\delta_s^*\) is the displacement thickness on the suction-side. These values are obtained from the curve-fits for a given \(Re_c\), \(\alpha^*\), and trip condition. In all these three equations, \(M_\infty\) stands for the free-stream Mach number, \(b\) is the span of the airfoil, \(\bar{D}_h\) is the directivity function, and \(H\) is the distance between the trailing edge and the receiver (microphone). The spectral representation of \(SPL_p\), \(SPL_s\), and \(SPL_\alpha\) are achieved by using the shape functions \(A\) and \(B\). These terms are functions of scaled Strouhal numbers written for each 1/3-octave band center frequency. The Strouhal Number for the pressure-side of the airfoil is written as:

\[
St_p = \frac{f \delta_p^*}{V_\infty}. \tag{4.5}
\]

Here, \(f\) is 1/3-octave band center frequency in Hz. For the suction side, the Strouhal Number is defined as

\[
St_s = \frac{f \delta_s^*}{V_\infty}. \tag{4.6}
\]
Figure 4.1: The 1/3-octave Sound Pressure Levels for validation case 7 ($\alpha^* = 1.5^\circ$, $V_\infty = 71.3$ m/s, and $Re_c = 1.497 \times 10^6$). Different components of the trailing edge noise spectra are calculated with the airfoil noise prediction method of Brooks et al.\textsuperscript{28}

The Strouhal numbers are scaled with $St_1$ or $St_2$, which are functions of the free-stream Mach number and the effective angle of attack. In Equations 4.1 and 4.2, $K_1$ is an amplitude factor, which is a function of the Reynolds number, $Re_c$. In Equation 4.1, $\Delta K_1$ is the level adjustment factor for the pressure-side contribution to the total trailing edge noise at non-zero angles of attack, and it is a function of the Reynolds Number based on the displacement thickness ($Re_{\delta^*}$) and the effective angle of attack. The amplitude factor $K_2$ used in Equation 4.3 is a function of $K_1$, the free-stream Mach number, and the effective angle of attack. The detailed expressions of the functions used to calculate different factors are given in Brooks et al.\textsuperscript{28}

The total trailing edge noise spectrum of each validation case was obtained using the airfoil noise prediction method described above. Since the acoustic measurements of the experiments were performed at a location 1.22 m away from the mid-span trailing edge where the directivity angles $\theta$ and $\phi$ were both 90$^\circ$, the same values were used in the sound pressure level prediction equations. In fact at this angular position, the directivity function $\bar{D}_h$ is equal to unity. Figure 4.1 shows the noise spectra obtained for validation Case 7 (Table 4.1) as an example. Each component obtained with Equations 4.1, 4.2, 4.3,
Table 4.2: The Overall Sound Pressure Level and the Noise Metric values obtained for the validation cases.

<table>
<thead>
<tr>
<th>Case (i)</th>
<th>$OASPL_i$ (dB)</th>
<th>$OASPL_{si}$ (dB)</th>
<th>$NM_i$ (dB)</th>
<th>$NM_{si}$ (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>71.72</td>
<td>0.00</td>
<td>53.36</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>55.72</td>
<td>-16.00</td>
<td>37.65</td>
<td>-15.71</td>
</tr>
<tr>
<td>3</td>
<td>56.82</td>
<td>-14.91</td>
<td>37.35</td>
<td>-16.01</td>
</tr>
<tr>
<td>4</td>
<td>60.18</td>
<td>-11.54</td>
<td>42.11</td>
<td>-11.25</td>
</tr>
<tr>
<td>5</td>
<td>66.56</td>
<td>-5.16</td>
<td>48.49</td>
<td>-4.87</td>
</tr>
<tr>
<td>6</td>
<td>71.78</td>
<td>0.06</td>
<td>53.15</td>
<td>-0.21</td>
</tr>
<tr>
<td>7</td>
<td>72.32</td>
<td>0.60</td>
<td>53.81</td>
<td>0.45</td>
</tr>
</tbody>
</table>

and the total spectrum calculated with Equation 4.4 can be seen in this Figure. The spectra of the noise components are symmetric with respect to their peak values. The maximum value of the total sound pressure level ($SPL_{TOT}$) is obtained at 1250 Hz for this case.

### 4.2.2 Calculation of the Overall Sound Pressure Level

Since the trailing edge Noise Metric developed here is a measure of total noise intensity, the relative change in the Noise Metric should be compared to the relative change in the Overall Sound Pressure Levels ($OASPL$) obtained from the experiments for each validation case. The $OASPL$ for a given case can be obtained by the summation of the 1/3-octave band $SPL_{TOT}$ values over all the frequencies in the spectrum:\(^\text{52}\)

$$OASPL(dB) = 10 \log \left[ \sum_{n=1}^{NMAX} 10^{0.1 \times SPL_{TOT}(f_n)} \right]$$  \hspace{1cm} (4.7)

where $f_n$ represents each 1/3-octave band frequency. $NMAX$ is the total number of frequencies. Note that since the summation is made in a logarithmic scale, the contributions to the $OASPL$ from the $SPL_{TOT}$ values that are far from the spectrum peak are negligible, as long as the spectrum is a bell shaped curve and has a single peak. For all the validation cases, the $OASPL$ values were obtained by performing the summation in Equation 4.7 over the 1/3-octave band frequencies between $f_1 = 31.5$ Hz and $f_{NMAX} = 40$ kHz, where the total number of frequencies is $NMAX = 32$. Ta-
Table 4.2 lists the Overall Sound Pressure Level values calculated for each validation case ($OASPL_i$). The third column of this table also gives the Scaled Overall Sound Pressure Value ($OASPL_{si}$) for each case,

$$OASPL_{si} = OASPL_i - OASPL_1 \quad (i = 1, \ldots, 7) \quad (4.8)$$

which a measure of the relative change from the reference value ($OASPL_1$) in $dB$.

### 4.3 Noise Metric Calculation

Two-dimensional CFD simulations were performed to calculate the value of Noise Metric for each validation case. The GASP code was used to obtain numerical solutions to the RANS equations. Menter’s SST $k$-$\omega$ model was used for turbulence modeling, and all viscous terms were retained in RANS equations. The CFD simulation of each validation case was performed with the assumption of fully turbulent flow. The reader should refer to Chapter 3 for the details of the physical modeling, the time integration, and the solution parameters used in two-dimensional CFD Simulations.

Each validation case was run with the corresponding experimental Reynolds Number $Re_c$ and the Mach Number given in Table 4.1. The physical chord dimensions used in the experiments were kept the same in the CFD simulations. Two structured grids were created for the computations. The grid for the NACA 0012 airfoil with a chord length ($c$) of 0.3048 m was used for Cases 1, 2, 4, 5, and 7 (Table 4.1) and the other grid prepared for the NACA 0012 airfoil with a chord length of 0.2286 m was used for Cases 3 and 6. The detailed description of the grids used in the validation studies can be found in Section 3.4.1.

For each validation case, the characteristic velocity ($u_0$) and length scale ($l_0$) for turbulence were extracted from the solution of the CFD simulations as described in Section A. Figure 4.2 shows the predicted turbulent kinetic energy and the length scale distribution along the normal direction at the upper surface trailing edge of NACA 0012 airfoil for validation Case 7 (Table 4.1). The characteristic velocity and the length scales obtained for the upper and the lower surface of the airfoil were then used in Equation 2.9 to calculate the Noise Metric (Equations 2.10 and 2.11). The other parameters used in the Noise
Metric calculation were the same as the experimental values: the distance to the receiver \((H)\) was taken as 1.22 m, the span length \(b\) was 0.4572 m, the trailing edge sweep \((\beta)\) was 0°, and both directivity angles \(\theta\) and \(\phi\) were 90° which made the directivity function \((\text{Equation 2.5})\) equal to unity. Table 4.2 lists the Noise Metric values calculated for each validation case \((NM_i)\). The last column of this table also gives the Scaled Noise Metric value \((NM_{si})\) for each case,

\[
NM_{si} = NM_i - NM_1 \quad (i = 1, \ldots, 7)
\]

which indicates the relative change from the reference value \((NM_1)\) in dB.

### 4.4 Validation Results

The summary of the Noise Metric validation approach is given in Figure 4.3. For each case, the experimental \(OASPL_i\) value is calculated from the 1/3-octave SPL spectrum, which is obtained using the semi-empirical airfoil noise prediction method described in Section 4.2.1. Then, the \(OASPL\) values are scaled with the reference case (case 1) to obtain the relative change in noise \((\text{Equation 4.8})\). The Noise Metric value of each validation case is also scaled in a similar way by using Equation 4.9.

Figure 4.4 shows the comparison of \(\overline{OASPL}_{si}\) and \(NM_{si}\) at each case. Here \(\overline{OASPL}_{si}\)
Validation Case $i$ ($i=1,..,7$)

Calculate Overall Sound Pressure Level, $OASPL_i$ (dB)

Calculate Noise Metric $NM_i$ (dB)

Calculate scaled $OASPL_i$ value, $OASPL_{si}$

Calculate scaled $NM_i$ value, $NM_{si}$

$OASPL_{si} = 10^{0.1 \times OASPL_{si}}$

$NM_{si} = 10^{0.1 \times NM_{si}}$

Figure 4.3: The steps followed in the Noise Metric validation study.

is defined as

$$OASPL_{si} = 10^{0.1 \times OASPL_{si}}$$

(4.10)

A similar definition can be made for $NM_{si}$:

$$NM_{si} = 10^{0.1 \times NM_{si}}$$

(4.11)

As can be seen from Figure 4.4, the agreement between the experiment and the predictions obtained with the Noise Metric is very good at various speeds, angles of attack, and Reynolds Number. This figure demonstrates that the new Noise Metric is capable of capturing the variations in the trailing edge noise as a relative noise measure when different flow conditions and parameters are changed. It can, therefore, be concluded that Noise Metric proposed here can be used to accurately judge the relative noise performance of clean wings as design and operating variables are changed.

Figure 4.4 also shows the noise predictions obtained with ANOPP. For each validation case, Overall Sound Pressure ($OASPL$) is calculated using the method described in Section 2.5. For relative comparison, $OASPL$ value of each case is scaled with the reference value obtained for case 1 using Equation 4.8. Results are presented for $OASPL_{si}$ variable, which is calculated with Equation 4.11. For all validation cases, relative change in the Noise Metric is closer to the change in experimental $OASPL$ values compared
Figure 4.4: The comparison of the Noise Metric predictions ($\overline{NM_{si}}$) and the $OASPL_{si}$ values obtained with ANOPP to the experimental $OASPL_{si}$ values of Brooks et al.\textsuperscript{28} at each NACA 0012 validation case.

to the ANOPP predictions. The agreement with the experimental data is significantly better for cases 2, 3, and 4. All these cases were obtained at relatively low Reynolds numbers (See Table 4.1). Since the clean wing noise formulation used in ANOPP does not take into account the effect of the Reynolds number on the turbulent kinetic energy (or turbulence intensity), the ANOPP predictions may not give accurate results when the Reynolds number differs significantly from the nominal value used in the derivation of the method. On the other hand, Noise Metric proposed in this study captures the variations in noise due to the change in Reynolds number, since the characteristic velocity and length scales are obtained from RANS simulations, which are performed with the exact Reynolds Number of the flow problem for each case.
Chapter 5

Two-Dimensional Parametric Noise Metric Studies

5.1 Need for Parametric Studies

The influence of the flight speed on the trailing edge noise is well-known, since the noise is proportional to the $5^{th}$ power of the velocity as shown by all the aeroacoustic theories on the subject. This effect is included in the proposed Noise Metric, since the characteristic velocity $u_0$ will change in proportion to the free-stream velocity in most cases. The Noise Metric also includes some parameters explicitly in the model like the directivity angles $\theta$ and $\psi$, distance to the observer $H$, and trailing edge sweep $\beta$. The functional dependency of trailing edge noise on these variables are again well described by the trailing edge theories.$^{36,20}$ However, in addition to the speed and other explicit parameters, one also would like to know the effect of the other variables used in aerodynamic optimization problems such as the lift coefficient and the wing geometry (thickness, airfoil shape, twist, etc.). The information obtained from parametric studies will provide valuable information about the effect of these variables on the trailing edge noise as predicted by the new Noise Metric. The experience gained in the parametric studies will also help to select the appropriate design parameters in building the response surfaces and increase the computational efficiency of optimization problems involving aerodynamic noise from a clean wing.
The parametric studies include both two- and three-dimensional cases. This chapter presents two-dimensional parametric Noise Metric studies. Two-dimensional parametric studies were done using two symmetric NACA Four-Digit airfoils (NACA 0012 and NACA 0009) and two supercritical (SC(2)-0710 and SC(2)-0714) airfoils. The results of the three-dimensional studies are presented in Chapter 6. A generic conventional transport wing was used for the three-dimensional studies.

5.2 Outline of the Two-Dimensional Studies

Two-dimensional parametric Noise Metric studies are presented under two groups depending on the geometry and the flow conditions used. The first group includes the studies done with the airfoils NACA 0012 and NACA 0009 at relatively low Reynolds Numbers. These studies were conducted with similar Mach and Reynolds Numbers to those used in the validation studies of Chapter 4. Although the Reynolds Numbers \( (Re_c = 1.497 \times 10^6 \text{ and } 1.837 \times 10^6) \) were low compared to the values that would be obtained with a typical transport aircraft at approach conditions, they were large enough to be able to model the flow as fully turbulent. The second group of two-dimensional studies were performed using two supercritical airfoils, SC2-0710 and SC2-0714, with the flow conditions at approach. For these cases, the CFD simulations were run with airfoils having realistic chord lengths that are comparable to the mean aerodynamic chord \((mac)\) of typical transport aircraft. The Reynolds Number for the supercritical airfoil cases was \( Re_c = 44 \times 10^6 \). The SC2-0710 and SC2-0714 airfoils can also be thought as more realistic aerodynamic geometries compared to NACA 0012 and NACA 0009 in the sense that modern transport aircraft have supercritical airfoil sections for most of their wing span to increase the cruise efficiency at transonic speeds. Both subsonic and supercritical airfoil results were obtained for subsonic speeds at a Mach number of 0.2, since the main interest of the Noise Metric study is the prediction of trailing edge noise at approach speeds. The main goal of the two-dimensional parametric studies is to investigate the effect of the maximum thickness ratio \((t/c)\) and the section lift coefficient \((C_l)\) on the noise as predicted by the current metric. This goal was studied in both subsonic and supercritical airfoil cases. The subsonic airfoil study also includes an example, which demonstrates the trailing edge noise reduction with \(C_l\) and \(t/c\) change.
5.3 Studies with NACA 0012 and NACA 0009 Airfoils

5.3.1 Geometry Description

The NACA 0012 and NACA 0009 airfoils (Figure 5.1) are among the most widely used geometries in experimental and numerical aerodynamic studies. Both NACA 0012 and 0009 are symmetric wing sections and belong to the same NACA Four-Digit airfoil family, but have different thickness values. The maximum thickness ratio \( t/c \) of NACA 0012 is 12% of its chord, whereas NACA 0009 has a \( t/c \) of 9%. The maximum thickness location for both airfoils is at 30% of the chord measured from the leading edge. Abbott and Doenhoff\(^{53}\) give an extensive description and analysis of NACA Four-Digit airfoils. In the same reference, the thickness distribution of the NACA four-digit sections is given by the following equation:

\[
\frac{z}{c} = \left(\frac{t}{c}\right) \left[ 1.4845 \sqrt{\frac{x}{c}} - 0.63 \left(\frac{x}{c}\right) - 1.7580 \left(\frac{x}{c}\right)^2 + 1.4215 \left(\frac{x}{c}\right)^3 - 0.5075 \left(\frac{x}{c}\right)^4 \right] \quad (5.1)
\]

The leading edge radius \( R_{le} \) is defined by

\[
\frac{R_{le}}{c} = 1.1019 \left(\frac{t}{c}\right)^2 \quad (5.2)
\]
Table 5.1: The NACA 0012 and NACA 0009 cases used to study the effect of $C_l$ and $t/c$ on the Noise Metric. For all the cases, $Re_c = 1.497 \times 10^6$, $Mach = 0.2$, $V_\infty = 71.3 \text{ m/s}$, $c = 0.3047 \text{ m}$, $\theta = 90^\circ$, $\psi = 90^\circ$, and $H = 1.22 \text{ m}$.

<table>
<thead>
<tr>
<th>NACA 0012</th>
<th>NACA 0009</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_l$</td>
<td>$C_d$</td>
</tr>
<tr>
<td>0.000</td>
<td>0.0112</td>
</tr>
<tr>
<td>0.162</td>
<td>0.0114</td>
</tr>
<tr>
<td>0.431</td>
<td>0.0124</td>
</tr>
<tr>
<td>0.643</td>
<td>0.0140</td>
</tr>
<tr>
<td>0.849</td>
<td>0.0164</td>
</tr>
<tr>
<td>1.046</td>
<td>0.0199</td>
</tr>
</tbody>
</table>

It can be seen from Equations 5.1 and 5.2 that the ordinate ($z$) of any point is directly proportional to the maximum thickness ratio. As a consequence of this, at each $x/c$ location, the thickness of NACA 0012 airfoil is %25 larger than the thickness of the NACA 0009 airfoil. It should also be noted that Equation 5.1 gives a finite thickness (0.252% of the chord) at the trailing edge. For the Noise Metric studies, after creating the airfoil geometry with Equations 5.1 and 5.2, the trailing edge was closed.

5.3.2 Test Cases

To study the effect of the section lift and the thickness ratio on the trailing edge noise, six, two-dimensional cases were computed for each airfoil at different lift values. Starting from zero lift, the angle of attack was increased for each case to obtain the Noise Metric at a different $C_l$ value. Table 5.1 gives an outline of the NACA 0012 and NACA 0009 cases used in this study. All computations were performed with the same Reynolds Number $Re_c = 1.497 \times 10^6$, $Mach = 0.2$, and free-stream velocity of $V_\infty = 71.3 \text{ m/s}$. Both NACA 0012 and NACA 0009 airfoils had a chord length of 0.3048 m. The details about the CFD Simulations and the grids used in the computations can be found in Chapter 3. For each case, the Noise Metric was calculated at a distance ($H$) 1.22 m away from the trailing edge with the directivity angles $\theta = 90^\circ$ and $\psi = 90^\circ$ (the same values used in the validation studies).

Figure 5.2 shows the section lift coefficient at different angles of attack and the drag polar
Figure 5.2: The section lift coefficient ($C_l$) vs. the angle of attack ($\alpha$) and the drag polars obtained for the NACA 0012 and NACA 0009 airfoils.

for both airfoils. As can be seen from this figure, the lift curve-slope $C_{l\alpha}$ is approximately the same for the NACA 0012 and the NACA 0009 airfoils. The lift curve-slope has a magnitude of 0.105 $deg^{-1}$ which is slightly less than the classical value of 0.110 $deg^{-1}$ ($2\pi rad^{-1}$) obtained with Thin Airfoil Theory. The variation from the theoretical value is due to the viscous effects. The computational lift curve-slope value is verified with the experimental data of Abbott and Doenhoff where $C_{l\alpha}$ is shown to be constant for both airfoils at various Reynolds Numbers. The highest section lift coefficient obtained in this study is 1.05 for both airfoils at $\alpha = 10^\circ$. The analysis of the experimental data indicates that the highest section lift coefficient obtained in this study is less than the stall value of each airfoil. For turbulent flow, the experimental data show that the maximum section lift coefficient ($C_{l_{\text{max}}}$) of the NACA 0012 airfoil is approximately 1.55 at $\alpha \approx 15^\circ$. NACA 0009 airfoil has a lower $C_{l_{\text{max}}}$ of 1.3 obtained at an angle of attack of approximately 12.5 degrees. As seen from the drag polar results, the section drag of the thicker airfoil is larger than the thinner one for lift coefficients up to 0.85. At this value, both airfoils have approximately the same section drag coefficient of $C_d = 0.0164$. Further increase in $C_l$ makes the drag of the thinner airfoil larger than the thicker one. This drag behavior is consistent with the experimental findings about the maximum lift values. For the thinner airfoil, boundary layer separation occurs at a lower maximum lift (or angle of attack) value. This implies that, after a certain angle of attack, the turbulent boundary layer on the thinner airfoil grows faster causing a larger increase in drag and eventually separation.
5.3.3 Effect of $C_l$ and $t/c$ on the Noise Metric

The total Noise Metric results computed at different section lift coefficient values are shown in Figure 5.3. The total Noise Metric is calculated using the pressure and the suction side contributions in Equation 2.11. At zero lift, a difference of 1.56 dB can be seen between the Noise Metric values of the NACA 0012 and NACA 0009 airfoils indicating the effect of the thickness. This difference remains approximately constant up to a $C_l$ of 0.2. Beyond this value, the Noise Metric of each airfoil starts to get larger with the increase of the lift coefficient. The increase in the Noise Metric is significant at higher lift coefficients. Between zero lift and the highest lift coefficient obtained in this study ($C_l = 1.05$), the difference in the noise is 8 dB for the NACA 0012 airfoil and 9 dB for NACA 0009. This shows the strong effect of the lift coefficient on the trailing edge noise. Although the noise of the thicker airfoil is larger than the thinner one for all lift coefficients covered in this study, the difference in the Noise Metric at each lift coefficient starts to get smaller when $C_l$ exceeds 0.43. At the highest lift coefficient, the difference eventually drops to 0.56 dB, which is 1 dB less than the value obtained at zero lift. This behavior of the Noise Metric is consistent with that of the drag coefficient discussed in the previous section. As the lift coefficient (or the angle of attack) increases, the Noise

![Figure 5.3: The total Noise Metric values obtained with NACA 0012 and NACA 0009 airfoils at different section lift coefficients ($Re_c = 1.497 \times 10^6$ and $Mach = 0.20$).](image-url)
Figure 5.4: The characteristic turbulent velocity ($u_0$) obtained for the suction side of NACA 0012 and NACA 0009 airfoils at different section lift coefficients ($Re_c = 1.497 \times 10^6$ and $Mach = 0.20$).

Metric of each airfoil starts to get larger, however this increase is more severe for the thinner airfoil due to the faster growth of the turbulent boundary layer. By examining the characteristic length ($l_0$) and the velocity scale ($u_0$) of the airfoil suction side (upper surface) in Figures 5.4 and 5.5, one can get a better understanding of the change in Noise Metric with the lift coefficient and the thickness. As can be seen from this figure, the characteristic velocity and the length scale increase with $C_l$ for both airfoils. However, the length scale of the thinner airfoil increases faster than that for the thicker one for lift coefficients that are greater than 0.65. At the highest lift coefficient value, the length scale of NACA 0009 is greater than the one obtained for NACA 0012. Beyond this lift coefficient value, one may expect to see the trailing edge noise of the NACA 0009 airfoil eventually becoming greater than that of the thicker airfoil.

The observations made on the NACA 0012 and NACA 0009 airfoils suggest that the trailing edge noise can be reduced to lower values by decreasing the lift coefficient. At lower lift coefficients, the thickness reduction also may also give noise benefits. Based on these observations, the noise reduction with $C_l$ and $t/c$ change is demonstrated with an example in the following section.
Figure 5.5: The characteristic length scale $l_0$ obtained for the suction side of NACA 0012 and NACA 0009 airfoils at different section lift coefficients ($Re_c = 1.497 \times 10^6$ and $Mach = 0.20$).

5.3.4 Noise Reduction with $C_l$ and $t/c$ change

The main purpose of the second study performed with NACA 0012 and NACA 0009 airfoils was to investigate the noise reduction by changing both the lift coefficient and the thickness ratio. To study this objective, the lift coefficient was reduced while increasing the chord length to have the same lift at a constant speed. Further reduction was sought by decreasing the thickness ratio, since lower noise levels can be obtained for the thinner airfoil in a certain range of lift coefficients as shown in the previous section. This two-dimensional study can be thought of as a simplified representation of increasing the wing area and reducing the overall lift coefficient of an aircraft at constant lift and speed.

As part of this study, three configurations were considered: (1) NACA 0012 airfoil with a chord length of 0.3048 m, (2) NACA 0012 airfoil with a chord length of 0.3741 m, and (3) NACA 0009 airfoil with a chord length of 0.3741 m. These configurations can be seen in Figure 5.6. All cases were run with a free-stream velocity ($V_\infty$) of 71.3 m/s and a Mach number of 0.2. The Reynolds number based on the chord ($Re_c$) was $1.497 \times 10^6$ for Case 1 and $1.837 \times 10^6$ for the other two cases. CFD simulations were performed
Figure 5.6: The NACA 0012 and NACA 0009 airfoils with different chord lengths used in the noise reduction study.

For each case. The Noise Metric was calculated at a distance \(H\) 1.22 m away from the trailing edge with the directivity angles \(\theta = 90^\circ\) and \(\psi = 90^\circ\). Note that these values are arbitrary, since the main interest of the study is in the relative change of the Noise Metric, and the receiver is assumed to be at the same location for all the cases.

Figure 5.7 shows the noise reduction history of this study. The starting point is Case 1 with the NACA 0012 airfoil at a lift coefficient \((C_l)\) of 1.046. \(C_l\) was reduced to 0.853 at Case 2 while increasing the chord length by 23% to keep the lift at a constant value of approximately 1010 Newtons. A noise reduction of 2.45 \(dB\) was achieved between Case 1 and Case 2. When the thickness of the airfoil was decreased by 25% \((NACA\ 0009)\) while keeping the same chord length and the lift, an additional reduction of 1.16 \(dB\) was obtained. Total noise reduction was 3.61 \(dB\). Decreasing the lift coefficient contributed 68% of the total noise reduction.
Figure 5.7: Noise metric reduction history obtained with NACA 0012 and NACA 0009 airfoils for various lift coefficients at constant lift.

This simple example showed that it is possible to reduce the trailing-edge noise by increasing the chord length (wing area) and decreasing the lift coefficient and the thickness ratio. Another benefit from this approach can come from eliminating the need to use high-lift devices which are the dominant airframe noise sources on approach. If sufficient lift can be obtained with an increased wing area without using high-lift devices, a significant reduction in noise may be achieved. However, all these changes should be performed in an MDO framework to account for the other aircraft design requirements.

5.4 Studies with SC(2)-0710 and SC(2)-0714 Airfoils

5.4.1 Geometry Description

The SC(2)-0710 and SC(2)-0714 airfoils were used in the second group of two-dimensional Noise Metric studies. These airfoils belong to the same supercritical airfoil family, but have different thickness ratios (Figure 5.8). The thickness ratio of the SC(2)-0710 airfoil is 10% of its chord, whereas SC(2)-0714 has a maximum thickness ratio of 14%. The maximum thickness is located at 37% of the chord for both airfoils. These airfoils were
The SC(2)-0710 and SC(2)-0714 airfoils. The airfoil coordinates are made non-dimensional with the chord length \(c\) and are not to scale.

developed to give a lift coefficient of 0.7 at design conditions.

The SC(2) family of supercritical airfoils was the result of successful aerodynamic research in NASA, which was directed toward developing practical airfoils with two-dimensional transonic turbulent flow and improved drag divergence Mach numbers while retaining acceptable low speed maximum lift and stall characteristics.\(^{56}\) The airfoils have typical geometric properties of supercritical airfoils, which are shaped to achieve supersonic flow with isentropic recompression. These geometric properties include a large leading edge radius, reduced curvature over the middle region of the upper surface, substantial aft camber, and thick trailing edges. For each airfoil, the design conditions were established by letting the Mach number float to determine the value required to achieve the target and the off-design pressure distributions at a specified maximum thickness and lift coefficient. The details of the development of SC(2)-0710 and SC(2)-0714 airfoils can be found in Harris.\(^{56}\)

The airfoil geometries were created with the data presented in Harris.\(^{56}\) Although the original geometries have thick trailing edges, the trailing edge of each airfoil was closed for the CFD simulations here.

It should be noted that both SC(2)-0710 and SC(2)-0714 airfoils are similar to the supercritical wing sections used in modern transport aircraft. The thicker airfoil can be thought as a representative inboard wing section, whereas the thinner one can be an example airfoil at an outboard station of a typical transport wing.
Table 5.2: The SC(2)-0714 and SC(2)-0710 cases used in the Noise Metric studies. For all the cases, $Re_c = 44 \times 10^6$, $Mach = 0.2$, $V_\infty = 68.0 \text{ m/s}$, $c = 9.54 \text{ m}$, $\theta = 90^\circ$, $\psi = 90^\circ$, and $H = 120 \text{ m}$.

<table>
<thead>
<tr>
<th>SC(2)-0714</th>
<th>SC(2)-0710</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_l$</td>
<td>$C_d$</td>
</tr>
<tr>
<td>0.550</td>
<td>0.0088</td>
</tr>
<tr>
<td>0.788</td>
<td>0.0098</td>
</tr>
<tr>
<td>1.245</td>
<td>0.0138</td>
</tr>
<tr>
<td>1.461</td>
<td>0.0169</td>
</tr>
<tr>
<td>1.665</td>
<td>0.0210</td>
</tr>
<tr>
<td>1.853</td>
<td>0.0263</td>
</tr>
<tr>
<td>2.020</td>
<td>0.0333</td>
</tr>
<tr>
<td>2.159</td>
<td>0.0432</td>
</tr>
</tbody>
</table>

5.4.2 Test Cases

The studies with supercritical airfoils were performed to investigate the effect of the lift coefficient and the thickness ratio on the Noise Metric at realistic flight conditions. CFD simulations were performed at a Reynolds Number of $Re_c = 44 \times 10^6$ with the free-stream

![Figure 5.9: The drag polars for SC(2)-0710 and SC(2)-0714 airfoils.](image-url)
velocity \( V_\infty = 68 \text{ m/s} \) and \( Mach = 0.2 \). These values approximately correspond to the conditions of a typical transport aircraft having a mean aerodynamic chord (\( mac \)) of 9.54 m at an altitude of 120 m before landing. At this location, the aircraft is approximately above the point where the noise certification measurements at approach are taken (See Figure 1.2 in Chapter 1). Consequently, the Noise Metric values were calculated for a receiver point where \( H = 120 \text{ m} \), \( \theta = 90^\circ \), and \( \psi = 90^\circ \).

Table 5.2 gives an outline of the cases used in noise studies with supercritical airfoils. The Noise Metric values were calculated at different lift coefficients. For each airfoil, the angle of attack was increased to get the highest lift coefficient before stall. The drag polar of each airfoil is shown in Figure 5.9. As can be seen from this figure, SC(2)-0710, the airfoil with the smaller thickness ratio has a lower maximum section lift coefficient value compared to the SC(2)-0714 airfoil. For lift coefficients greater than 0.8, the drag of the SC(2)-0710 airfoil is larger at the same lift. This drag behavior is consistent with the qualitative observations made in the studies with NACA airfoils. For these subsonic airfoils, after a certain lift coefficient, the drag of the thinner airfoil exceeds the drag values of the thicker airfoil. (See Section 5.3.2). As a result of this, the thinner subsonic airfoil has a lower maximum lift coefficient, which is also observed for the thinner supercritical airfoil of this study.

### 5.4.3 Characteristic Velocity and Length Scales

Before the discussion of the Noise Metric results, one may gain useful insight into the behavior of the trailing edge noise by examining the characteristic velocity (\( u_0 \)) and the length scale (\( l_0 \)) at different lift coefficient values. Figures 5.10 and 5.11 show the characteristic velocity and length scale at the trailing edge of each airfoil as a function of the lift coefficient. The results are presented both for the suction and the pressure sides of the airfoils. At the lowest lift coefficient for each airfoil, characteristic velocity of the pressure side is comparable to the suction side value. As the lift coefficient increases, the suction side velocity remains approximately constant up to \( C_l = 1.35 \). For \( C_l < 1.35 \), the characteristic velocity of the thicker airfoil is slightly larger. For \( C_l > 1.35 \), a substantial increase in the characteristic velocity can be noticed for both airfoils, and this change is larger for the thinner airfoil. The characteristic velocity on the pressure side of each airfoil decreases continuously as the lift coefficient increases. The magnitude of pressure
Figure 5.10: Characteristic turbulent velocity ($u_0$) obtained at the trailing edge of SC(2)-0710 and SC(2)-0714 airfoils at different section lift coefficients ($Re_c = 44 \times 10^6$ and $Mach = 0.20$).

Figure 5.11: Characteristic length scale ($l_0$) obtained at the trailing edge of SC(2)-0710 and SC(2)-0714 airfoils at different section lift coefficients ($Re_c = 44 \times 10^6$ and $Mach = 0.20$).
side velocity is slightly larger for the SC(2)-0714 airfoil compared to the SC(2)-0710 with a constant margin of approximately 0.4 m/s.

An interesting observation can be made for the length scale behavior in Figure 5.11 at relatively low lift coefficients (between $C_l \approx 0.5$ and 0.95). For this range of $C_l$, the length scale on the pressure side of each airfoil is larger than the suction side value. As the lift coefficient increases, the length scale at the trailing edge of the pressure side decreases continuously in a similar trend observed for the characteristic velocity. The length scale of the suction side gets larger as the lift coefficient increases. This change is significant and much larger for the thinner airfoil at higher lift coefficients. The increase in the length scale at the trailing edge of the suction side is related to the rapid growth of boundary layer due to the strong adverse pressure gradients. The effect of the adverse pressure gradient on the suction side can also be seen from the skin friction ($C_f$) values at the trailing edge of the airfoils (Figure 5.12). As the lift coefficient increases, the magnitude of the skin-friction drops for both airfoils. The positive skin friction values indicate that two-dimensional boundary layer separation does not occur for any of the cases. However, the sharp reduction in the skin friction values, which approach to zero at higher lift coefficients suggest the influence of the adverse pressure gradient on the

![Figure 5.12](image)

Figure 5.12: The skin friction ($C_f$) values obtained at the trailing edge of the suction side of SC(2)-0710 and SC(2)-0714 airfoils for different section lift coefficients.
Figure 5.13: The turbulent kinetic energy (TKE) and the length scale $l_0$ profiles at the upper surface trailing edge of SC(2)-0710 and SC(2)-0714 airfoils for various section lift coefficients. The filled symbols show the maximum TKE and the corresponding length scale values.

The turbulent kinetic energy increases as the lift coefficient increases. This is because the adverse pressure gradient close to the upper surface trailing edge increases the thickness of the turbulent boundary layer and the magnitude of the turbulent fluctuations. These values become larger as the flow gets closer to separation. It should also be noted that, starting from $C_l \approx 1.2$, the change in the skin friction magnitude with respect to the lift coefficient ($-\partial C_f/\partial C_l$) is larger for the thinner airfoil. This explains why the maximum lift coefficient of the SC(2)-0710 is less than the value observed for the SC(2)-0714 airfoil. The larger drop in the skin-friction magnitude of the thinner airfoil can be regarded as an indicator of earlier boundary layer separation.

Figure 5.13 gives a good description of how the turbulent kinetic energy and the length scale change as the lift coefficient increases. This figure shows the suction side turbulent...
kinetic energy and the length scale profiles at the trailing edge of SC(2)-0710 and SC(2)-
0714 airfoils for selected lift coefficients. The characteristic length scale used in the Noise
Metric calculations corresponds to the length scale at the location where the maximum
\( TKE \) value is obtained (See Chapter 2.3). As can be seen from Figure 5.13, turbulent
kinetic energy profiles vary smoothly with the lift coefficient. Each \( TKE \) profile has a
maximum value which gets larger with the increase of lift coefficient, and the location
of the maximum \( TKE \) moves away from the wall as the lift coefficient increases. After
reaching the maximum value, the turbulent kinetic energy starts to decrease and eventu-
ally vanishes at a certain distance from the wall. This distance also gets larger with the
increase of the lift coefficient, which indicates that the the boundary layer gets thicker.
Examining the length scale profiles, one can see that the length scale is zero on the wall
and it starts to increase until a certain value is reached. Beyond this location, the length
scale remains approximately constant through a certain region. One may assume that
the region where the length scale increases corresponds to the inner part of the boundary
layer, and the constant values are obtained in the outer region. These observations are
consistent with the results of many eddy viscosity turbulence models,\textsuperscript{57} which produce
increasing length scales as a function of wall distance in the inner region and constant
values for the outer part of the boundary layer in wall-bounded turbulent flows. As can
be seen from Figure 5.13, the maximum value of the turbulent kinetic energy is located
approximately at a point where the length scale starts to remain constant. This may
imply that the maximum \( TKE \) values shown in Figure 5.13 are located in the outer
region of the boundary layer at the specified lift coefficients. One may also see this from
Figure 5.14, which shows the velocity profiles at the suction side trailing edge of the
SC(2)-0714 airfoil for \( C_l = 0.550 \) and \( C_l = 1.665 \). The velocity profiles are presented
with inner variables \( z_n^+ \) and \( u^+ \) where
\[
  u^+ = \frac{U}{u_*} \tag{5.3}
\]
In addition to the profiles obtained from CFD simulations, a velocity profile for turbulent
flow on a flat plate with zero pressure gradient has also been calculated at the same
Reynolds Number, \( Re_x = 44 \times 10^6 \) where \( x = 9.54 \text{ m} \). The inner region velocity profile
for zero pressure gradient was constructed using the wall functions defined for the viscous
sub-layer and logarithmic region.\textsuperscript{58} For the outer region, the wake function introduced
by Coles\textsuperscript{59} was used. The skin-friction (\( C_f \)) was calculated using the formulation by
Figure 5.14: Velocity profiles at the trailing edge of the SC(2)-0714 airfoil at different section lift coefficients. The zero pressure gradient case is calculated with theoretical predictions at $Re_c = 44 \times 10^6$ and shown only for qualitative comparison. Red dashed-line shows the $TKE_{max}$ location for $C_l = 0.550$, and black dashed-line marks the $TKE_{max}$ location for $C_l = 1.665$. Schultz-Grunow given in Schetz. The zero pressure gradient case is presented only for the purpose of qualitative comparison. One can see from Figure 5.14 that, compared to the zero pressure gradient case, the velocity profiles at the trailing edge of the SC(2)-0714 airfoil have much larger wake regions indicating the effect of the adverse pressure gradient on the mean velocity profile. For the zero pressure gradient case, the skin-friction is higher, which gives a lower magnitude of $u^+$ at the edge of the boundary layer. Examining the airfoil cases only, one can see that the profile obtained for $C_l = 1.665$ has a larger wake region. This demonstrates the fact that increasing the lift coefficient enlarges the size of wake region. In Figure 5.14, the dashed-lines mark the $TKE_{max}$ locations obtained for $C_l = 0.550$ and 1.665. Both points are in the outer (wake) region of the turbulent boundary layer. The maximum TKE of the $C_l = 1.665$ case is located further away from the wall compared to the lower $C_l$ case. For a zero pressure gradient turbulent flow on a flat plate with $Re_x = 4.2 \times 10^6$, Klebanoff showed that the maximum
turbulent kinetic energy reaches its maximum value very close to the wall, approximately 1% of the boundary layer thickness. This indicates that, in the absence of pressure gradients, the maximum turbulent kinetic is located in the inner region of the boundary layer. The Reynolds number of the current study \( (Re_c = 44 \times 10^6) \) is rather high in the sense that it is very difficult to find experimental data on turbulent boundary layer flows with adverse pressure gradients at such Reynolds Numbers. However, at moderately large Reynolds Numbers, a number of experimental studies are available. Among these, experimental work by Krogstad et al.\cite{61,62} show the influence of strong adverse pressure gradient on the turbulence structure in a boundary layer. Their data show that the mean velocity profiles of the adverse pressure gradient flow are dominated with an extensive wake region as observed in the CFD simulations of the current study. They also show significant differences in the turbulent kinetic energy budget between the zero and the adverse pressure gradient flow. Their data demonstrate that, for the adverse pressure gradient flow, the maximum value of the kinetic energy appears in the outer region, approximately 45% of the boundary layer thickness.

5.4.4 Noise Metric Predictions

After calculating the characteristic length and velocity scales, the Noise Metric can be obtained both for the suction and the pressure sides of the airfoils using Equations 2.9 and 2.10. The total Noise Metric is calculated with Equation 2.11 using the suction and the pressure side values. Figures 5.15 and 5.16 show the predicted suction side, pressure side, and the total Noise Metric for the SC(2)-0710 and the SC(2)-0714 airfoils at different lift coefficients. Examining these figures, one can see that the contribution of the pressure side to the total noise is significant at lower lift coefficients. For SC(2)-0714 airfoil at \( C_l = 0.55 \), the pressure side contribution is even greater than the suction side. For both airfoils, as the lift coefficient increases, the suction side Noise Metric gets larger while the pressure side component starts to decrease. At lift coefficients greater than \( C_l \approx 1.2 \), the suction side dominates the total noise, and the contribution of the pressure side is negligible. The Noise Metric change observed for the suction and the pressure sides are consistent with the observations made for the behavior of the characteristic velocity and the length scale in the previous section.
Figure 5.15: The Noise Metric values obtained for the suction and pressure sides of the SC(2)-0710 airfoil. The total Noise Metric value is obtained using Equation 2.11.

Figure 5.16: The Noise Metric values obtained for the suction and pressure sides of the SC(2)-0714 airfoil. The total Noise Metric value is obtained using Equation 2.11.
Figure 5.17: The Comparison of total Noise Metric values obtained with SC(2)-0710 and SC(2)-0714 airfoils.

Figure 5.17 shows the comparison between the total Noise Metric values. The noise of each airfoil remains approximately constant up to a certain lift coefficient value. At this range of lower $C_l$, the thicker airfoil has higher Noise Metric values. The difference is approximately 2 dB at $C_l = 0.7$. A dramatic increase in the Noise Metric value can be observed for each airfoil at higher lift coefficients. The large increase in the Noise Metric at high lift coefficients originates from the increase of both the maximum $TKE$ and the characteristic length scale $l_0$ on the trailing edge of the suction side. Figure 5.17 shows another important effect of the thickness ratio on the Noise Metric. The noise from the SC(2)-0710 airfoil is greater than the noise of the thicker airfoil at $C_l > 1.35$ due to the larger increase in the characteristic velocity and length scale (See Figures 5.10 and 5.10). This implies that reducing the thickness ratio may in fact increase the noise at higher lift coefficients.

When the Noise Metric results of the supercritical airfoils are compared to those obtained with the four digit symmetric NACA airfoils, a number observations can be seen to be common for both studies: (1) an approximately constant Noise Metric value is found at lower lift coefficients, (2) a large increase in the Noise Metric is observed after a certain
lift coefficient value, (3) less trailing edge noise is predicted for the thinner airfoil at lower lift coefficients, and (4) larger (or a tendency to become larger as observed for NACA 0009 airfoil) trailing edge noise is indicated for the thinner airfoil at higher lift coefficients. These are all general qualitative observations. The quantitative details, such as the range of lift coefficients where the Noise Metric remains constant and the rate of increase of the Noise Metric with $C_l$ at higher lift coefficients differ between two airfoil classes studied. The supercritical airfoils have substantial geometric differences compared to the NACA four digit airfoils. For example, they have large aft camber close to the trailing edge, whereas subsonic airfoils are symmetric. The thickness ratio of subsonic and supercritical airfoils are also different. Although both NACA four digit airfoil and supercritical airfoil studies were performed with a fully turbulent flow assumption, the large difference between the Reynolds Number of each study ($Re_c = 1.497 \times 10^6$ for NACA airfoils and $Re_c = 44 \times 10^6$ for the supercritical airfoil cases) can also another factor creating the quantitative differences.

Figure 5.18 gives a comparison between the Noise Metric results of the SC(2)-0714 airfoil and the $OASPL$ values predicted with the formula derived by Lockard and Lilley$^{10}$ (Equation 2.17 in Chapter 2). The predictions obtained with ANOPP$^{27}$ are also included to emphasize the fact that $OASPL$ values calculated with this method do not change with the lift coefficient as explained in Section 2.5. Both the Noise Metric and the $OASPL$ values are scaled with their corresponding values at $C_L = 0.550$ so that a comparison in the relative change can be made. As can be seen from this figure, both the Noise Metric and the approximation by Lockard and Lilley capture the increase in noise as $C_l$ increases. However, there are differences in the actual values at each lift coefficient, which may be due the different airfoil geometry (NACA 4412) used in the derivation of Lockard and Lilley’s model (See Section 2.4).
Figure 5.18: Comparison between the scaled total Noise Metric value ($NM_s$) of SC(2)-0714 airfoil and the scaled Overall Sound Pressure Level ($OASPL_s$) obtained with the formula by Lockard and Lilley\textsuperscript{10} (Equation 2.17), and ANOPP.\textsuperscript{27}
Chapter 6

Three-Dimensional Parametric Noise Metric Studies

The two-dimensional Noise Metric studies supplied valuable information about the effect of the section lift coefficient and the thickness ratio on the trailing edge noise. Besides the experience gained from two-dimensional studies, it was also desirable to perform the parametric studies on a finite wing to include the three-dimensional effects. One of the objectives was to examine the effect of the overall lift coefficient on the clean wing airframe noise by using a realistic wing geometry at realistic conditions. This study also enabled the investigation of the spanwise variation of the characteristic velocity and length scale as the lift coefficient was changed.

The three-dimensional studies presented in this chapter were performed with two versions of a conventional transport wing at realistic approach conditions. The initial Noise Metric results were obtained for various lift coefficient values using the baseline wing geometry which had a baseline twist distribution. After the evaluation of the results obtained with the baseline wing geometry, it was seen that a possible noise reduction might be achieved by modifying the wing twist distribution. To study this objective, a modified wing with a new twist distribution was generated. All the other geometric parameters were kept the same as their original values. A vortex lattice code was used to find the desired twist distribution that would give a possible reduction in the trailing edge noise. The Noise Metric calculations were performed again for various lift coefficients using the modified wing geometry.
This chapter first presents the details about the wing geometry and the test conditions used in the CFD simulations. Next, the results obtained with the baseline geometry including the discussion about the spanwise variation of the characteristic velocity and the length scales are given. Before the presentation of the results with the modified wing geometry, the details of the new twist distribution are described. The modified wing results are compared to the baseline wing data to investigate the effect of the twist distribution on the trailing edge noise and to verify if the desired level of noise reduction was achieved.

6.1 Description of the Baseline Wing Geometry

The baseline geometry used in the three-dimensional studies is the Energy-Efficient Transport (EET) Wing. This is a generic conventional transport aircraft wing used in many experimental studies at NASA. For this work, the original dimensions of the experimental model was scaled so that the mean aerodynamic chord is 9.54 m, which is comparable to that of a Boeing-777 like airplane. The scaled wing has a planform area ($S_{\text{ref}}$) of 511 m$^2$ and a wing span ($b$) of 64.6 m. The planform view of the wing is shown in Figure 6.1. The EET wing consists of an inboard and an outboard section. The outboard section starts at $\eta_b = 0.375$ ($\eta = 2y/b$). The geometric details of the wing are presented in Table 6.1. Note that the planform area includes the leading and the trailing edge extensions in the inboard section. Unless specified with a subscript, all lift and drag coefficients ($C_L$ and $C_D$) presented in this study use $S_{\text{ref}}$ for scaling. The trapezoidal wing area, which does not include the leading and the trailing edge extensions in the inboard section is 83% of the planform area. Therefore, $C_L$ and $C_D$ should be multiplied with a factor of 1.205 to get the lift and drag coefficients ($C_{L_{\text{tp}}}$ and $C_{D_{\text{tp}}}$) that use the trapezoidal wing area as the reference scaling value.

The wing airfoil sections were defined at twenty spanwise ($\eta$) stations, and the data describing the upper and the lower surface of each airfoil are tabulated in Jacobs. The baseline wing sections have thick trailing edges. For the CFD simulations, the trailing edge of each airfoil has been closed. The airfoils with closed trailing edges were used in the construction of the wing upper and lower surfaces. The wing sections between each defined airfoil station were obtained with straight-line wrapping. The wing tip was
Table 6.1: EET wing geometry parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean aerodynamic chord ($mac$)</td>
<td>9.54 m</td>
</tr>
<tr>
<td>Span ($b$)</td>
<td>64.6 m</td>
</tr>
<tr>
<td>Wing break location ($\eta_b$)</td>
<td>0.375</td>
</tr>
<tr>
<td>Wing planform area (including leading and trailing edge extensions in the inboard section) ($S_{ref}$)</td>
<td>511 $m^2$</td>
</tr>
<tr>
<td>Trapezoidal wing planform area (excluding leading and trailing edge extensions in the inboard section) ($S_{tp}$)</td>
<td>424 $m^2$</td>
</tr>
<tr>
<td>Aspect ratio ($AR = b^2/S_{ref}$)</td>
<td>8.16</td>
</tr>
<tr>
<td>Taper ratio (based on trapezoidal wing planform)</td>
<td>0.41</td>
</tr>
<tr>
<td>Quarter chord sweep angle ($\Lambda_{c/4}$)</td>
<td>30.0°</td>
</tr>
<tr>
<td>Inboard trailing edge sweep angle ($\Lambda_{TEi}$)</td>
<td>0.0°</td>
</tr>
<tr>
<td>Outboard trailing edge sweep angle ($\Lambda_{TEo}$)</td>
<td>24.24°</td>
</tr>
<tr>
<td>Dihedral angle</td>
<td>5.0°</td>
</tr>
<tr>
<td>$t/c$ at the root ($t/c$ location in the chord)</td>
<td>14.4% (32%)</td>
</tr>
<tr>
<td>$t/c$ at the break ($t/c$ location in the chord)</td>
<td>12.2% (36%)</td>
</tr>
<tr>
<td>$t/c$ at the tip ($t/c$ location in the chord)</td>
<td>10.4% (36%)</td>
</tr>
</tbody>
</table>

![EET Wing Planform](image)
Figure 6.2: The airfoils used in the root, break, and the tip location of the EET wing. The airfoil coordinates are made non-dimensional with the local chord length \( c \) and are not to scale.

Figure 6.3: The spanwise variation of the maximum thickness ratio \( t/c \) and the actual maximum thickness \( t \).
rounded for efficient grid generation. Figure 6.2 shows the wing sections at the root, break, and the tip stations of the wing. This figure shows that the airfoils at the break and tip location have the geometric properties of a typical supercritical wing section with a large leading edge radius, significant aft camber, and a fairly flat upper surface. The root airfoil has a significant amount of aft camber, however it has a more curved upper surface after the maximum thickness location. Figure 6.3 shows the thickness ratios at various spanwise stations. A smooth variation of the actual maximum thickness along the span can also be seen. Figure 6.4 shows the baseline twist ($\theta_b$) distribution of the baseline wing geometry. At the wing root, the twist has a positive value of 1.79°. At the break location the twist angle is reduced to $-0.25^\circ$, and at the tip, it is equal to $-3.06^\circ$ indicating the washout at the outboard section. The twist distribution between the root and the tip does not vary as a single smooth function but should be treated separately in the inboard and the outboard section as can be seen from Figure 6.4. A modified wing was obtained by changing the twist distribution to seek reduction in the trailing edge noise. The details of this study are described in Section 6.4 of this chapter. It should be noted that all the other geometric parameters were the same between the baseline and the modified wing.

Figure 6.4: The baseline twist distribution ($\theta_b$) of the baseline wing.
6.2 Test Conditions

For the CFD simulations and the Noise Metric calculations, the same test conditions were used both for the baseline and the modified wing cases. These are the same parameters used for the supercritical airfoil cases in Chapter 5 and correspond to the approach conditions of a typical transport aircraft. The CFD simulations were performed with a free-stream velocity of \( V_\infty = 68 \text{ m/s} \) which corresponds to \( Mach = 0.2 \). The Reynolds number based on the mean aerodynamic chord was \( Re_{mac} = 44 \times 10^6 \). The Noise Metric was evaluated at an altitude of 120 m, for an observer at the ground level directly below the aircraft which corresponds to \( \theta = 90^\circ \) at \( y = 0 \) plane (see Figure 2.3). The azimuthal angle \( \psi \) was calculated at each spanwise station, however the effect of the change in \( \psi \) along the span was negligible for this altitude.

6.3 Baseline Wing Results

6.3.1 Lift and Drag Characteristics

The baseline wing calculations were performed at eight different angles of attack ranging from 0° to 14° with increments of 2°, and Figure 6.5 shows the lift coefficients and corresponding wing loading \( (W/S) \) values obtained at each angle of attack. The \( C_L \) vs. \( \alpha \) curve is linear up to 12°, where \( C_L = 1.084 \). The overall lift coefficient at \( \alpha = 0^\circ \) \( (C_{L0}) \) is equal to 0.219 for this wing. At the last angle of attack, one can see the break of the linear pattern which indicates stall. This can also be seen from the drag polar given in Figure 6.6. The sharp increase in drag at the last angle of attack, where \( C_L = 1.106 \), is due to a massive flow separation on the outboard section of the wing. Table 6.2 gives an outline of the results obtained with the baseline wing.

With this wing configuration, the highest lift loading \( (L/S_{ref}) \) value that could be achieved was 315.7 \( kg/m^2 \) (64.8 \( lb/ft^2 \)). On the other hand, for a B-777 like transport aircraft, it was found that wing loading \( (W/S_{ref}) \) is approximately 432 \( kg/m^2 \) (88.8 \( lb/ft^2 \)) when using the maximum design landing weight of such an aircraft and \( S_{ref} \) (planform area) of the current wing geometry. Since the aircraft is at a 3° glide slope on approach, the lift should be equal to %99.8 of the weight of the aircraft for a steady
Table 6.2: The baseline wing cases of the three-dimensional Noise Metric studies. For all the cases, \( Re_c = 44 \times 10^6 \), \( Mach = 0.2 \), \( U_\infty = 68 \) m/s, \( mac = 9.54 \) m, \( \theta = 90^\circ \), and \( H = 120 \) m. Note that \( C_L \) is the lift coefficient calculated based on the wing planform area \( (S_{ref}) \), and \( C_{L_{tp}} \) is the lift coefficient that uses the trapezoidal wing area \( (S_{tp}) \) as the reference scaling value.

<table>
<thead>
<tr>
<th>( C_L )</th>
<th>( C_{L_{tp}} )</th>
<th>( C_D )</th>
<th>( C_{D_{tp}} )</th>
<th>( NM (dB) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.375</td>
<td>0.452</td>
<td>0.0152</td>
<td>0.0183</td>
<td>38.35</td>
</tr>
<tr>
<td>0.534</td>
<td>0.643</td>
<td>0.0215</td>
<td>0.0259</td>
<td>38.19</td>
</tr>
<tr>
<td>0.689</td>
<td>0.831</td>
<td>0.0309</td>
<td>0.0373</td>
<td>39.19</td>
</tr>
<tr>
<td>0.836</td>
<td>1.007</td>
<td>0.0424</td>
<td>0.0510</td>
<td>40.94</td>
</tr>
<tr>
<td>0.970</td>
<td>1.169</td>
<td>0.0560</td>
<td>0.0674</td>
<td>44.76</td>
</tr>
<tr>
<td>1.084</td>
<td>1.306</td>
<td>0.0724</td>
<td>0.0872</td>
<td>51.08</td>
</tr>
<tr>
<td>1.106</td>
<td>1.332</td>
<td>0.1263</td>
<td>0.1522</td>
<td>64.42</td>
</tr>
</tbody>
</table>

flight. Since the glide angle is very small, this flight condition can still be considered as level flight. Although one can reach relatively high lift coefficients with a clean wing by increasing the angle of attack without having substantial separation, it is clear that it would be almost impossible to achieve the lift required to sustain a conventional aircraft with a typical wing area and speed at the approach without using a traditional high-lift system or an innovative high-lift concept such as a jet flap\textsuperscript{24} or circulation control device.\textsuperscript{63} This again points to the importance of having a large wing area to reduce the wing loading at a constant speed, if one wants to design a clean wing at approach conditions that can fly with a low lift coefficient. As a consequence of this, lowering the lift coefficient will reduce the trailing edge noise as will be shown later in this chapter. Also, a lower lift coefficient that would not require high-lift devices by the virtue of an increased wing area will give further noise reduction with the elimination of the noise from the high-lift devices.

The section lift coefficient \( (C_l) \) and the spanload distributions of the baseline wing are given in Figures 6.7 and 6.8 for different overall lift coefficient \( (C_L) \) values. The spanload and the \( C_l \) exhibit smooth variations along the half-span for all \( C_L \) except the highest value obtained from the case with massive flow separation at the outboard of the wing. The section lift coefficient stays at relatively lower values in the vicinity of the root. At the highest overall lift coefficient \( (C_L = 1.106) \), the \( C_l \) at the root is approximately 0.9, which
Figure 6.5: Overall lift coefficient ($C_L$) and Lift Loading ($L/S_{ref}$) vs. angle of attack ($\alpha$) for the baseline wing.

Figure 6.6: The drag polar of the baseline wing.
is 64% of the maximum section lift coefficient value. Starting from the root, the section lift coefficient increases until it reaches its maximum value ($C_{l_{\text{max}}}$). For $0.219 \leq C_L \leq 0.689$, the $C_{l_{\text{max}}}$ is located between $\eta = 0.5$ and $\eta = 0.6$ stations. Beginning from $C_L = 0.836$, the maximum section lift coefficient moves to 38% of the half span, which is very close the wing break point ($\eta_b = 0.375$) and then stays at this location for the higher $C_L$. From Figure 6.9, one can see that the maximum section lift coefficient increases as a linear function of the overall lift coefficient, except for the last $C_L$ value obtained at the stall condition. In the linear range, the $C_{l_{\text{max}}}/C_L$ ratio remains approximately constant with $C_{l_{\text{max}}}$ being 25% higher than the overall lift coefficient value. Figure 6.7 shows that, starting from the wing break point, the change in $C_l$ is very small up to $\eta \approx 0.65$. At higher $C_L$, one may expect to see the start of flow separation in this region due to the existence of high section lift coefficient values. In fact, at $C_L = 1.106$, the sudden drop in $C_l$ due to flow separation is located in this region. For the same $C_L$, a large loss in lift at the outboard section of the wing can also be seen from the spanload distribution given in Figure 6.8.

Figure 6.10 shows the skin-friction ($C_f$) contours on the wing upper surface at three overall lift coefficient values. The skin friction values presented in this picture were obtained using the component of the wall shear stress in the free-stream direction on each cell face of the wing upper surface. Therefore, a negative skin friction value indicates that the free-stream component of the wall shear is in the opposite direction to the main flow direction. For the cases without massive flow separation, negative values of the skin friction at the trailing edge region may be interpreted as the indicators of incipient flow separation. For $C_L = 0.375$, the skin friction iso-lines show a smooth pattern along the span except for the small kink at the break point. At this $C_L$, the skin friction values remain positive on the entire wing surface including the trailing edge-region. Both the trend of the iso-lines and the positive value of the skin friction indicate no flow separation. For $C_L = 0.970$, at the outboard of the wing, a small distortion in the iso-line pattern can be seen starting from $\eta \approx 0.5$. This may indicate that, at this lift coefficient, the three-dimensional effects start to become important on the wing boundary layer flow. For the same lift coefficient, the skin friction values approach very small values ($\sim 10^{-4}$) at the trailing edge region between the wing break location and $\eta \approx 0.9$. For higher lift coefficients ($C_L > 0.970$), the skin-friction drops to negative values at the trailing edge of the outboard wing indicating the incipient flow separation. Although the incipient
Figure 6.7: Section lift coefficient ($C_l$) distributions for the baseline wing.

Figure 6.8: Spanload distributions for the baseline wing.
flow separation that occurs very close to the trailing edge does not have a significant effect on the lift characteristics and general pressure field on the wing, its influence on the trailing edge noise may be important, since the change in the maximum turbulent kinetic energy and the characteristic length scale become significant when the flow is close to separation as seen in Chapter 5. Massive separation on the outboard section of the wing is visible at $C_L = 1.106$. Due to the limitations on predicting complex flows with tree-dimensional separation using eddy viscosity turbulence models, the quantitative results obtained for this case may not be highly accurate. However the results may still give useful information about the qualitative nature of the problem.

6.3.2 Characteristic Velocity and Length Scale Results

Examining the behavior of the maximum turbulent kinetic energy ($TKE_{max} = u_0^2$) and the characteristic length scale ($l_0$) along the span of a finite wing may be useful to understand the change in the trailing edge noise as $C_L$ increases. Figures 6.11 and 6.12 show the spanwise variations of the $TKE_{max}$ and $l_0$ for the baseline wing at five $C_L$ values. At $C_L = 0.534$, $TKE_{max}$ remains approximately constant along the span except
Figure 6.10: Skin friction contours on the upper surface of the baseline wing at different $C_L$ values.
in the tip region, where the maximum turbulent kinetic energy increases due to the tip vortex. An increase in the length scale can also be seen in the same region. Except in the tip region, the characteristic length scale is largest at the root section and decreases along the span. This may be explained with the argument that the length scale follows the chord length distribution along the span. In the absence of strong adverse pressure gradients, one may expect to see a thicker boundary layer at the trailing edge of the root section, since it has the largest chord length. In a thicker boundary layer, the characteristic length scale is also expected to be larger. Similar observations on $TKE_{max}$ and $l_0$ can be made for lift coefficients that are less than 0.836. At these relatively low $C_L$ values, the three-dimensional effects are minimum, since $TKE_{max}$ and length scale do not change significantly in the spanwise direction. The three-dimensional effects become important at higher lift coefficients ($C_L \geq 0.836$). By examining Figures 6.11 and 6.12, one can see a significant increase in $TKE_{max}$ and $l_0$ starting from $C_L = 0.836$ especially on the outboard section of the wing where the section lift coefficients are higher. Since the section lift coefficients remain at relatively low values in the inboard section, the change in the turbulent kinetic energy and length scale is not as large as the one observed for the outboard section. These results are consistent with the previous observations made on the skin friction change on the upper surface of the wing. The skin friction values approaching zero at the trailing edge of the outer wing imply the increase in $TKE_{max}$ and the length scale. At $C_L = 1.084$ which corresponds to $\alpha = 12^\circ$, a large separation region is not observed but incipient flow separation at the trailing edge of the outboard wing can still cause significant increase in $TKE_{max}$ and $l_0$. The change is more dramatic when there is massive flow separation as observed at the highest lift coefficient.

At higher lift coefficient values, the maximum $TKE$ and $l_0$ are not uniform along the span, and they get larger at outboard sections due to three-dimensional effects. This shows the importance of calculating the Noise Metric, especially at high lift coefficients, with a characteristic velocity and length scale that vary along the span. It also points to the fact that using an average value for the characteristic velocity or the length scale along the span will not work at higher lift coefficients, simply because the noise metric is proportional to the integration of $\overline{u_0^5} \times \overline{l_0}$ term along the span and $\overline{u_0^5} \times \overline{l_0}$ is not equal to $\overline{u_0^5} \times \overline{l_0}$ in general (Here the overbar is used to represent the average along the span).

Figure 6.13 shows the turbulent kinetic energy contours in the vicinity of the wing tip trailing edge region at various lift coefficients. The last case with massive flow separation
Figure 6.11: Maximum $TKE \ (u_0^2)$ distributions along the upper surface trailing edge of the baseline wing.

Figure 6.12: Characteristic length scale ($l_0$) distributions along the upper surface trailing edge of the baseline wing.
Figure 6.13: Turbulent Kinetic Energy contours in the vicinity of the baseline wing tip trailing edge region (looking from downstream) at different $C_L$ values. Note that the maximum TKE of the last case ($C_L = 1.106$) is much greater than the contour upper limit.
is shown to demonstrate the large increase in the $TKE$ in a qualitative way. Looking at the other cases with no significant flow separation, one can see the turbulent kinetic energy field associated with the tip vortex. As the lift coefficient increases, the size of the turbulent kinetic energy region gets larger and the location of the maximum $TKE$ moves outwards in the spanwise direction. Although both $TKE$ and the length scale increase in the tip region, this does not cause a significant change in the magnitude of the total trailing edge Noise Metric, since the spanwise extent of the tip region is small. This result is verified by the experiments of Brooks and Marcolini which showed that the contribution of the tip-vortex-formation noise to the overall broadband noise spectrum was much less than the contribution of the turbulent-boundary-layer-trailing-edge noise at various angles of attack for a three-dimensional NACA 0012 airfoil section.

### 6.3.3 Noise Metric Results

After calculating the characteristic length and the velocity scales along the span, the Noise Metric was obtained both for the upper and the lower surfaces of the baseline wing using Equations 2.9 and 2.10. The total Noise Metric was calculated with Equation 2.11 using the upper and the lower surface values. Figure 6.14 shows the upper surface, lower surface, and the total Noise Metric values for the baseline wing at different lift coefficients. In addition to $C_L$, the $C_{L_{tp}}$ values are also shown in the abscissa of this Figure. As defined before, $C_L$ is the lift coefficient calculated based on the wing planform area ($S_{ref}$), and $C_{L_{tp}}$ is the lift coefficient that uses the trapezoidal wing area ($S_{tp}$) as the reference scaling value. Examining Figure 6.14, one can see that, at lower lift coefficient values ($C_L \leq 0.534$), the Noise Metric remains approximately constant, and the contribution to the total noise from the lower surface is significant at these low $C_L$ values. As the lift coefficient increases, the upper surface starts to dominate the noise, and the Noise Metric gets larger. Recall that a similar behavior in the contribution of the upper and lower surfaces to the total noise was observed for the supercritical airfoils used in two-dimensional studies (See Section 5.4.4). When $C_L$ exceeds 0.534, a gradual increase in the Noise Metric can be noticed. At $C_L = 0.970$, the Noise Metric becomes 6.6 $dB$ higher than the value obtained at $C_L = 0.534$. Note that in the skin friction results for $C_L \leq 0.970$, no flow separation was observed on the upper surface of the wing. However, for $0.970 < C_L < 1.106$, mild separation at the trailing edge of the outboard wing was
Figure 6.14: Noise metric values obtained with the baseline wing at different lift coefficient values. In the abscissa, $C_L$ stands for the lift coefficient calculated based on the wing planform area ($S_{ref}$), and $C_{Ltp}$ is the lift coefficient that uses the trapezoidal wing area ($S_{tp}$) as the reference scaling value.

detected. For this range of $C_L$, a larger increase in the noise can be seen. The change in the trailing edge noise becomes 12.7 $dB$ between $C_L = 0.375$ and $C_L = 1.084$. At the highest lift coefficient, a dramatic increase in the Noise Metric due to the massive flow separation can be noticed.

The Noise Metric results of a generic conventional transport wing show that, in order to get the maximum reduction in the trailing edge noise of a clean wing for a given lift and approach speed, the overall lift coefficient should be lowered so that it is in the upper limit of the region where the Noise Metric is constant and has the minimum value. Further reduction of $C_L$ in the constant Noise Metric range would not give any significant noise benefits. If the results of the current study are used as representative values in an example case, one can see that reducing $C_L$ from 0.836 to 0.534 will give a noise reduction of 2.8 $dB$. If it can be assumed that the wing weight does not change with the increase of the wing area in an ideal situation, a 36% reduction in the lift coefficient will require a proportional increase in the wing area for the same approach speed. In reality, increasing
the wing area will make the wing heavier mainly because of the structural constraints. Due to the increase in the aircraft weight, the required $C_L$ for the increased wing area will be higher than its ideal value. As a result of this, the noise reduction will be less than the maximum possible value obtained in the ideal case. However, the difference between the ideal noise reduction and the real case may be small depending on the Noise Metric region where the final $C_L$ falls into. Note that increasing the wing area may also increase the noise at the same lift coefficient, since the characteristic length and velocity scales may get larger due to the increase of the turbulent boundary layer thickness at the trailing edge. Since many of the parameters that change the trailing edge noise also affect the other aircraft design requirements, the noise reduction studies should be performed in the context of a Multidisciplinary Design and Optimization framework. Finally, the above discussions are all based on the assumption that the initial wing area or the speed is large enough to sustain the aircraft without a conventional high-lift system so that the dominant noise source is the wing trailing edge at the approach. A Blended-Wing-Body configuration with a large wing area may be a good example for such an aircraft.

6.4 Effect of the Twist on Noise

The results of the baseline wing study suggest that the twist distribution, especially at the outboard section of the wing, may play an important role in the reduction of the trailing edge noise at high lift coefficients. It has been seen that the characteristic length scale and maximum turbulent kinetic energy become larger on the outboard section of the wing at higher lift coefficients. High section lift coefficient values on the outboard wing make the boundary layer flow over this region more likely to separate due to the increase of the adverse pressure gradients at higher $C_L$. By modifying the twist on the outboard section of the wing, it may be possible to lower the section lift coefficient values and delay the increase in $TKE$ and $l_0$. In order to keep the same total lift value, the section lift coefficients at the inboard wing should be increased. Since the inboard wing has relatively low section lift coefficients, a moderate increase of the twist in this region should not create a significant change in the noise contribution to the total value from the inboard section. To study the possible noise reduction by changing the twist, a modified wing with a new twist distribution was generated. All the other geometric parameters were kept the same as the baseline wing.
In order to avoid the computational expense of finding a candidate twist distribution with RANS simulations, a vortex panel code was used. The section lift coefficient and the spanload distributions obtained with the vortex panel code were evaluated to find the desired twist distribution that would give a possible reduction in the trailing edge noise. The following section describes the method used in obtaining the new twist distribution. The comparison of the modified wing results with the baseline wing data is given next to investigate the effect of the twist distribution on the trailing edge noise and to verify if the desired level of noise reduction has been achieved.

### 6.4.1 Modified Twist Distribution

The method used to modify the twist distribution utilizes VLMpc vortex lattice code developed by John Lamar at NASA Langley Research Center. It has been widely used at Virginia Tech for aircraft preliminary design and aerodynamic case studies. The program can treat two lifting surfaces using up to 200 panels. In the code, vortex flows are estimated using the leading edge suction analogy. Although VLMpc is a low-fidelity aerodynamics tool, it can give good predictions of the section lift, spanload, induced drag, and the overall lift coefficient for different wing configurations in non-complex flows.

For the studies with VLMpc code, the wing planform was modeled with 13 panels in the spanwise direction and 6 panels in the streamwise direction (Figure 6.15). Before changing the twist, VLMpc code was tested for estimating the section lift distribution of the baseline wing at $C_L = 0.970$. The original twist angles specified at 13 spanwise panel locations were used as input to the code. Figure 6.16 shows the comparison of the VLMpc results with the section lift distribution obtained from GASP. Despite small differences in the root and the wing break regions, there is good agreement between the results of each code.

To try to reduce the trailing edge noise, a generic twist distribution was proposed. This twist distribution is defined with three twist angles specified at the root, wing break, and the tip. The twist angle at the wing break point is kept the same as the original value. The twist angles at the inboard stations are calculated by assuming straight line wrap between the root and the wing break. Similarly, the twist angles for the outboard sections were obtained using the straight line wrap assumption between the wing break and the tip. In order to get a possible reduction in the trailing edge noise, the twist angles for the
Figure 6.15: The wing planform with vortex lattice locations used in VLMpc code.

Figure 6.16: The section lift comparison for the baseline wing at $C_L = 0.970$. 
Figure 6.17: The baseline and the modified twist distributions.

Outboard wing should be lowered to reduce the section lift coefficients. In other words, more washout should be given to the outboard section. This can be done by specifying a tip angle that is less than the original value. A reduction in the tip twist angle must increase the root incidence, since the overall lift is kept the same. Calculation of the root twist is done iteratively. For a given tip twist angle, the root twist is increased until the required lift is obtained with a specified angle of attack. In this study, the specified angle of attack was obtained using the VLMpc code with the baseline twist distribution and the same required lift coefficient. Other values could be used for the specified angle of attack, however for each angle of attack, a different root twist value will be obtained. At each iteration, the VLMpc code outputs the angle of attack at which the required lift is obtained with the given twist distribution. The root twist (thus the twist distribution) is updated until the VLMpc code gives the specified angle of attack value.

For noise reduction purposes, it is desirable to reduce the tip angle as much as possible. However while doing so, it should be kept within a certain range, since reducing the tip angle beyond acceptable limits can cause negative lift in the tip region at low angles of attack and create a huge drag penalty. In addition to this, since the total lift should be kept constant, over-reduction of the tip twist may push the root incidence to a high value.
that is not acceptable. Considering these facts, a final twist distribution that could give a possible noise reduction was created using the procedure described above (Figure 6.17). The required lift coefficient was 0.970 and the specified angle of attack was 12.45° in the design procedure. The new distribution has a twist angle of −6.61° at the tip and 3.48° at the root. The twist at the wing break is kept at its original value, which is −0.25°.

Figure 6.18 shows the modified and the baseline wing section lift distributions at $C_L = 0.970$. Both distributions are obtained using the VLMpc code. One can see that the new twist distribution creates the desired effect on the section lift distribution. On the outboard of the wing, the section lift values are reduced. At the inboard region, the lift coefficients are increased. The maximum section lift coefficient of the modified wing seems to be slightly higher than the baseline one. It should also be noted that the purpose of using VLMpc code in the twist study was to be able to see a proper change in section lift distribution for possible noise reduction. High fidelity results including the Noise Metric values are obtained from the RANS simulations performed with the modified wing geometry.
Table 6.3: The modified wing cases of the three-dimensional Noise Metric studies. For all the cases, \(Re_c = 44 \times 10^6\), \(\text{Mach} = 0.2\), \(U_\infty = 68 \text{ m/s}\), \(mac = 9.54 \text{ m}\), \(\theta = 90^\circ\), and \(H = 120 \text{ m}\). Note that \(C_L\) is the lift coefficient calculated based on the wing planform area \((S_{\text{ref}})\), and \(C_{Ltp}\) is the lift coefficient that uses the trapezoidal wing area \((S_{tp})\) as the reference scaling value.

<table>
<thead>
<tr>
<th>(C_L)</th>
<th>(C_{Ltp})</th>
<th>(C_D)</th>
<th>(C_{Dtp})</th>
<th>(NM) (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.375</td>
<td>0.452</td>
<td>0.0149</td>
<td>0.0179</td>
<td>38.79</td>
</tr>
<tr>
<td>0.574</td>
<td>0.692</td>
<td>0.0259</td>
<td>0.0312</td>
<td>38.48</td>
</tr>
<tr>
<td>0.730</td>
<td>0.879</td>
<td>0.0363</td>
<td>0.0437</td>
<td>38.72</td>
</tr>
<tr>
<td>0.827</td>
<td>0.997</td>
<td>0.0443</td>
<td>0.0534</td>
<td>39.85</td>
</tr>
<tr>
<td>0.966</td>
<td>1.164</td>
<td>0.0582</td>
<td>0.0702</td>
<td>42.72</td>
</tr>
<tr>
<td>1.064</td>
<td>1.283</td>
<td>0.0705</td>
<td>0.0850</td>
<td>45.75</td>
</tr>
<tr>
<td>1.133</td>
<td>1.365</td>
<td>0.0811</td>
<td>0.0977</td>
<td>48.90</td>
</tr>
<tr>
<td>1.178</td>
<td>1.420</td>
<td>0.1159</td>
<td>0.1397</td>
<td>60.21</td>
</tr>
</tbody>
</table>

6.4.2 Comparison with the Baseline Wing Results

After a new twist distribution was determined using the VLMpc code, a modified wing was constructed. The same grid parameters and dimensions of the baseline wing were used to create a new computational grid around the modified geometry. Using the same flow conditions and physical modeling parameters of the baseline wing cases, three-dimensional RANS simulations were performed at various overall lift coefficients. For each run, the angle of attack was increased to vary the lift coefficient. The lift-curve slope \((C_{L\alpha})\) of the modified wing was predicted using a number of lift coefficients from the results of the CFD simulations at specified angles of attack. With the \(C_{L\alpha}\) information, two of the modified wing cases \((C_L = 0.375\) and \(C_L = 0.966\) cases) were obtained at approximately the same lift coefficients \((C_L = 0.375\) and \(C_L = 0.970\)) as two of the baseline wing cases. These two cases are used for the comparison of the maximum turbulent kinetic energy \((TKE_{max})\) and the length scale \((l_0)\) distributions to get a better understanding of the Noise Metric results at low and high lift coefficients. Table 6.3 gives the lift, drag, and the Noise Metric results obtained with the modified wing. Since the results of the baseline geometry supplied detailed information about the change of the section lift, spanload, \(TKE_{max}\) and \(l_0\) distributions as well as the Noise Metric with the lift coefficient, the discussion of the current section focuses on the effect of the twist
Figure 6.19: Comparison between the overall lift coefficient ($C_L$) and the wing loading ($L/S_{ref}$) values of the baseline and the modified wing at various angles of attack.

Figure 6.19 shows the overall lift coefficient values of the baseline and the modified wing at various angles of attack. The lift-curve of the modified wing is slightly shifted up due to the change of the twist. The lift-curve slopes of the baseline and the modified wing are approximately the same at the linear range. With a least-squares fit to the $C_L$ data in the linear range, the $C_{L_0}$ of the baseline wing has been found to be $0.075 \, \text{deg}^{-1}$. For the modified wing, $C_{L_0}$ has been calculated as $0.074 \, \text{deg}^{-1}$. The extrapolation of the lift-curve to zero degrees angle of attack gives a lift coefficient ($C_{L_0}$) of 0.280 for the modified wing. The $C_{L_0}$ of the baseline wing is 0.219, which is 78% of the modified wing value. The lift coefficients obtained at the highest angle of attack ($\alpha = 14.0^\circ$) are located beyond the linear range of the lift-curve for both wings. At this angle of attack, both wings have massive flow separation as seen from the skin-friction contours. The size of the separation region is larger for the baseline wing. Therefore, the drag rise associated with the flow separation is larger for the baseline wing as can be observed from the drag polar results shown in Figure 6.20. Excluding the cases with massive flow separation, one can see that the drag of the modified wing is higher than that of the
baseline wing at the same lift. The difference in drag gets larger for $C_L$ values up to 0.75 and starts to decrease for $C_L > 0.75$. One plausible reason for the increase of the drag difference may be the larger increase in the induced drag of the modified wing because of the less ideal spanload distribution. At low lift coefficients, the induced drag will be less for both wings. One may also think that, for $C_L > 0.75$, the increase in the form drag of the baseline wing becomes larger resulting in a decrease in the drag difference. It can be seen that the modified wing is not necessarily a better wing aerodynamically at moderate lift coefficients, since it has higher drag values. However, it may give better noise performance at high lift coefficients as will be shown next.

The Noise Metrics for the baseline and the modified wing are compared in Figure 6.21. At low lift coefficients ($C_L < 0.6$), there is no significant difference in the Noise Metric, so the change in the twist distribution does not seem to affect the noise behavior at low lift coefficients. Starting from $C_L \approx 0.6$, the difference between the baseline and the modified wing Noise Metric values can be noticed. As the lift coefficient increases, the trailing edge noise gets larger for both wings, however the noise increase is more severe for the baseline wing. It can be seen that the modified twist reduces predicted the noise.
Figure 6.21: The comparison between the total Noise Metric values of the baseline and the modified wing at different lift coefficient values.

level at higher lift coefficients. Figure 6.21 shows that the goal of trailing edge noise reduction with twist change was achieved for high lift coefficients.

Table 6.4 gives a comparison between the drag and the Noise Metric values of the baseline and the modified wing at three selected lift coefficient values. At $C_L = 0.375$, the Noise Metric of the modified wing is 0.44 $dB$ higher than the baseline wing value. This difference is considered to be small when compared with the higher lift results in a decibel scale.

At the same lift coefficient, the drag difference is only 3 counts. A larger difference in the Noise Metric can be observed for the higher lift coefficients. At $C_L = 0.970$, the noise of the modified wing is 2 $dB$ less than the baseline wing, and a 4 $dB$ noise reduction was achieved at $C_L = 1.064$. Although the modified wing gives significant noise benefits at higher lift coefficients, it has larger drag values. At $C_L = 0.970$, the drag penalty is 22 counts which corresponds to a 4% increase in the drag. This example again demonstrates a necessity of performing the noise reduction study in an MDO framework to obtain a configuration with optimum noise and aerodynamic characteristics.

The effect of the twist on the Noise Metric at low and high lift coefficients can be understood better by examining the maximum turbulent kinetic energy and the length scale.
Table 6.4: Comparison between the drag and the Noise Metric values of the baseline and the modified wings at selected lift coefficients

<table>
<thead>
<tr>
<th>$C_L$</th>
<th>$C_D$</th>
<th>$NM$ (dB)</th>
<th>$C_D$</th>
<th>$NM$ (dB)</th>
<th>$\Delta C_D$</th>
<th>$\Delta NM$ (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.375</td>
<td>0.0152</td>
<td>38.35</td>
<td>0.0149</td>
<td>38.79</td>
<td>-0.0003</td>
<td>0.44</td>
</tr>
<tr>
<td>0.970</td>
<td>0.0560</td>
<td>44.76</td>
<td>0.0582</td>
<td>42.72</td>
<td>0.0022</td>
<td>-2.04</td>
</tr>
<tr>
<td>1.064</td>
<td>0.0691</td>
<td>49.73</td>
<td>0.0705</td>
<td>45.75</td>
<td>0.0014</td>
<td>-3.98</td>
</tr>
</tbody>
</table>

distributions. Figures 6.22 and 6.23 show the $TKE_{max}$ and the $l_0$ distributions of the baseline and the modified wing at $C_L = 0.375$. At this lift coefficient, the $TKE_{max}$ distributions are almost the same regardless of the twist used. The length scales of the modified wing are slightly larger in the inboard section and slightly lower in the outboard section compared to the baseline wing values. These observations indicate that the twist does not create a significant change on $TKE_{max}$ and $l_0$ at relatively low lift coefficients. Therefore, no significant difference is observed in the Noise Metric at these lift coefficients.

At $C_L = 0.970$, a large difference can be noticed in $TKE_{max}$ and $l_0$ distributions in the outboard section (Figures 6.24 and 6.25). The modified twist reduces the maximum turbulent kinetic energy and the length scale significantly in this region. Since the increase in $TKE_{max}$ and $l_0$ in the inboard section of the modified wing is not as severe as the level of reduction at the outboard section, a 2 $dB$ total noise reduction can be achieved. This can also be seen from Figure 6.26, which shows the inboard and the outboard contributions to the total noise for the baseline and the modified wing at $C_L = 0.970$. Although there is 0.5 $dB$ increase in the inboard section contribution with the modified twist, a 3.8 $dB$ reduction is achieved in the outboard section contribution. The large benefit in the outboard contribution gives a 2 $dB$ total noise reduction. Figure 6.26 also shows that the baseline wing has a larger contribution to the total noise from its outboard section. The contributions to the total noise from the inboard and the outboard sections become comparable with the twist change.

On the outboard section of the modified wing, the lower $TKE_{max}$ and $l_0$ values are

---

\footnote{At $C_L = 1.064$, the Noise Metric and the drag coefficient of the baseline wing are obtained from curve fits to the data.}
Figure 6.22: Maximum $TKE (u'^2)$ distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.375$.

Figure 6.23: Characteristic length scale ($l_0$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.375$. 
Figure 6.24: Maximum $TKE$ ($u_0^2$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.970$.

Figure 6.25: Characteristic length scale ($l_0$) distributions along the upper surface trailing edge of the baseline and the modified wing at $C_L = 0.970$. 
obtained with the reduction of the section lift coefficients (Figure 6.27). Since the lift is
kept at the same value, the loss of lift on the outboard section must be balanced with the
increase on the inboard section (Figure 6.28). The section lift coefficients have relatively
lower values close to the root section at the inboard, therefore slight changes in their
magnitudes do not create a large noise penalty. It has also been seen that the onset
of incipient separation near the outboard section trailing edge of the baseline wing has
been moved to a higher \( C_L \) value with the reduction of the section lift values on the
outboard section with the modified twist (Figure 6.21). The massive separation occurs
also at a higher lift coefficient value for the modified wing. Overall, it can be said that
the modified twist delays the increase in \( TKE \) and \( l_0 \) at higher lift coefficients, and as a
result of this a significant noise reduction is achieved.

In Figure 6.29, the Noise Metric results for the baseline and modified wings are com-
pared to the \( OASPL \) predictions obtained with the formula by Lockard and Lilley\(^{10}\)
(Equation 2.17). The predictions obtained with ANOPP\(^{27}\) are again included to show
that \( OASPL \) values calculated with this method do not change with the lift coefficient
(See Section 2.5). The Noise Metric results and the \( OASPL \) values are scaled with their
corresponding values at \( C_L = 0.375 \) to be able to compare the relative change. In Lil-
ley and Lockard’s formulation, scaled \( OASPL \) values were calculated separately with
the overall lift coefficient (\( C_L \)) and the maximum section lift coefficient (\( C_{l_{\text{max}}} \)) of the
baseline wing (see Figure 6.9) using Equation 2.17. Since the increase in the trailing
edge noise at higher overall lift coefficients is due to the large increase in the length scale
and \( TKE_{\text{max}} \) at spanwise stations with high section lift coefficients, it is expected that
replacing \( C_L \) with \( C_{l_{\text{max}}} \) in the formula by Lockard and Lilley (Equation 2.17) could give
better noise predictions. At lower lift coefficients (\( C_L < 0.5 \)), the agreement between all
noise measures is good. As \( C_L \) increases, the difference between the scaled Noise Metric
and the \( OASPL_s \) predicted by Lockard and Lilley gets larger. The \( OASPL_s \) calculated
with \( C_{l_{\text{max}}} \) is closer to the Noise Metric results, however the difference is still significant
at high lift coefficients. One possible reason for the difference at high lift values may be
the fact that the prediction by Lockard and Lilley uses two-dimensional airfoil calcula-
tions to model the \( C_L \) effect on the noise. Therefore, their model may not capture the
three-dimensional effects which increase the turbulent kinetic energy and the length scale
by significant amounts at high lift coefficients.
Figure 6.26: Contributions to the total Noise Metric from the inboard and the outboard sections of the baseline and the modified wing at $C_L = 0.970$.

Figure 6.27: Section lift coefficient ($C_l$) distributions for the baseline and the modified wing at $C_L = 0.970$. 
Figure 6.28: Spanload distributions for the baseline and the modified wing at $C_L = 0.970$.

Figure 6.29: The comparison between the scaled total Noise Metric values ($NM_s$) and the scaled Overall Sound Pressure Levels ($OASPL_s$) obtained with ANOPP,\textsuperscript{27} and with the formula by Lockard and Lilley\textsuperscript{10} (Using both $C_L$ and $C_{l_{\text{max}}}$ in Equation 2.17).
Chapter 7
Discussion and Conclusions

7.1 Summary of the Results

The objective of this study was to model airframe noise from a clean wing at approach conditions in a way that could be used in MDO studies. A new Noise Metric was developed that may be used for optimization problems involving aerodynamic noise from a clean wing. The Noise Metric is not the exact value of the noise intensity, however it was shown to be an accurate indicator as a relative trailing edge noise measure. It can be calibrated to give the exact noise signature value with available experimental data and flight measurements. The Noise Metric modeling approach uses the theory of Ffowcs Williams and Hall\textsuperscript{36} as the starting point. The final form of the Noise Metric includes characteristic velocity and length scales that are obtained from three-dimensional, steady, RANS simulations with a two equation $k$-$\omega$ turbulence model. One unique feature of the proposed Noise Metric is the modeling of the length scale, which is directly related to the turbulent structure of the flow at the trailing edge. The current Noise Metric model is also capable of capturing three dimensional effects which become important at high lift coefficients, since the characteristic velocity and the length scales are allowed to vary along the span of the wing. The new Noise Metric has been formulated so that it can capture the effect of different design variables on the clean wing airframe noise such as the aircraft speed, lift coefficient, and wing geometry.

Noise metric validation was performed with seven test cases that were selected from a two-
dimensional NACA 0012 experimental database. The agreement between the experiment and the predictions obtained with the new Noise Metric was very good at various speeds, angles of attack, and Reynolds Number, which showed that the Noise Metric is capable of capturing the variations in trailing edge noise as a relative noise measure when different flow conditions and parameters are changed.

Parametric studies were performed to investigate the effect of different design variables on the noise as indicated by the Noise Metric. The information obtained from these studies not only contributes to the general knowledge in the field, but also helps the selection of the appropriate design parameters with respect to their importance in the actual design process.

Two-dimensional parametric studies were done using two four-digit NACA airfoils (NACA 0012 and NACA 0009) and two supercritical (SC(2)-0710 and SC(2)-0714) airfoils. NACA four-digit airfoil studies were performed with relatively low Reynolds Numbers ($Re_c = 1.497 \times 10^6$ and $1.837 \times 10^6$), whereas the supercritical airfoil cases were evaluated at a Reynolds number of $Re_c = 44 \times 10^6$ which was obtained using a realistic chord length that was comparable to the mean aerodynamic chord of a typical transport aircraft at approach conditions.

The main goal of the two-dimensional parametric studies was to investigate the effects of the maximum thickness ratio ($t/c$) and the section lift coefficient ($C_l$) on the Noise Metric. Examining the Noise Metric results for the supercritical and the NACA four-digit airfoils, a number of common observations could be made: (1) an approximately constant Noise Metric value is found at lower lift coefficients, (2) a large increase in the Noise Metric occurs after a certain lift coefficient value, (3) less trailing edge noise is indicated for the thinner airfoil at lower lift coefficients, and (4) larger (or a tendency to become larger as observed for the NACA 0009 airfoil) trailing edge noise is found for the thinner airfoil at higher lift coefficients. Two dimensional studies also showed that the large increase in the Noise Metric at high lift coefficients originates from the increase of both the maximum turbulent kinetic energy and the length scale at the trailing edge of the suction side. An example study with the NACA 0012 and NACA 0009 airfoils demonstrated a reduction in the trailing edge noise by decreasing the thickness ratio and the lift coefficient, while increasing the chord length to keep the same lift at a constant speed. This 2-D study can be considered as a simplified representation of increasing the wing area and reducing the overall lift coefficient of an aircraft at constant lift and speed.
The three-dimensional studies were performed with two versions of a conventional transport wing (EET Wing) at realistic approach conditions. The first wing geometry had a baseline twist distribution. After the evaluation of the results obtained with the baseline wing geometry, it was seen that a possible noise reduction could be achieved by modifying the wing twist distribution. To study this objective, a modified wing with a new twist distribution was generated. All the other geometric parameters were kept the same as their original values. The modified twist distribution was designed so that it would reduce the section lift values at the outboard sections of the wing while increasing the spanload at the inboard sections to keep the same lift.

Similar to the results obtained from two-dimensional cases, the Noise Metric studies with the baseline and the modified wing showed that the trailing edge noise remains constant at low lift coefficients ($C_L < 0.6$). The contribution to the total noise from the lower surface is significant at these low $C_L$ values. As the lift coefficient increases, the upper surface starts to dominate the noise, and the total Noise Metric gets larger. The change in the Noise Metric is significant when there is incipient separation at the trailing edge. The change is more dramatic for cases with massive flow separation. The wing cases also showed that, at higher lift coefficient values, the maximum $TKE$ and $l_0$ are not uniform along the span, and they get larger on outboard sections due to three-dimensional effects. This shows the importance of calculating the Noise Metric, especially at high lift coefficients, with a characteristic velocity and length scale that vary along the span.

It has been observed that the wing with the modified twist is not necessarily a better wing based on its aerodynamic performance at moderate lift coefficients, since it has higher drag values compared to the original wing. However, it gives better noise performance at high lift coefficients ($C_L > 0.6$). The comparison with original wing results showed that the twist change does not affect the maximum turbulent kinetic energy ($TKE_{max}$) and the length scale ($l_0$) distributions along the span at low lift coefficients. As a result of this, no significant difference is observed between the Noise Metric values for the baseline and the modified wing. At high lift coefficients, the modified twist reduces the noise by keeping the $TKE_{max}$ and $l_0$ values lower at the outboard section of the wing. The reduction can be as much as 4 $dB$ at a lift coefficient of $C_L = 1.064$. With the modified twist, the onset of incipient separation was delayed compared to the baseline wing, and the massive separation was observed at a higher maximum lift value.
7.2 Implications of the Results for Design

The results obtained in this study show the importance of the lift coefficient \( C_L \) on the airframe noise of a clean wing. In order to reduce the noise level, a wing should be designed so that it can give the required lift at a lower lift coefficient value. Using the Noise Metric results of the current study, one can see that, in order to get the maximum reduction in trailing edge noise of a clean wing for a given lift and approach speed, the overall lift coefficient should be lowered so that it is in the upper limit of the region where the Noise Metric is constant and has the minimum value. Further reduction of \( C_L \) in the constant Noise Metric range would not give any significant noise benefits. If the speed of the aircraft is kept constant, only an increase in the wing area will reduce the lift coefficient at constant lift. The results of the current study show that although one can reach relatively high lift coefficients with a clean wing by increasing the angle of attack without having substantial separation, it would be almost impossible to achieve the lift required to sustain a conventional aircraft with a typical wing area and speed on approach without using a traditional high-lift system or an innovative high-lift concept such as a jet flap or a circulation control device. This again points to the importance of having a larger wing area if the required lift is to be obtained without high-lift devices. A lower lift coefficient that would not require high-lift devices by the virtue of an increased wing area will give further noise reduction with the elimination of the high-lift related noise. As can be seen from the results of two-dimensional studies, thinner wing sections may also give some noise reduction at low lift coefficients. Although twist change can give significant noise benefits at higher lift coefficients, its effect will be very little at lower lift values. It should be noted that increasing the wing area and using a thinner wing section may increase the wing weight at some occasions. Therefore, the required lift could be more than the original value. Since the weight penalty should be balanced with an increase in the lift coefficient or speed, the noise reduction may be less than its intended value. A twist distribution designed to reduce noise may also create a drag penalty. All these examples show that many of the parameters that change the trailing edge noise also affect the other aircraft design requirements. Therefore, noise reduction studies should be performed in an MDO framework to find the optimum configuration.
References


Appendix A

Extracting Characteristic Velocity and Length Scales from CFD Simulations

The RANS simulations with Menter’s $k$-$\omega$ SST turbulence model were performed to calculate the turbulent characteristic velocity ($u_0$) and the turbulence length scale ($l_0$) using GASP, which is a three-dimensional, structured, finite-volume, RANS code. The details of the CFD simulations are given in Chapter 3. The first step in the calculation procedure is to obtain the $TKE$ and $\omega$ profiles at the trailing edge of an airfoil or wing from the solution of the CFD simulations:

- For two-dimensional (airfoil) problems, the $TKE$ and $\omega$ profiles for the upper and the lower surfaces are constructed using the values extracted from the cell center locations normal to the airfoil surface at the trailing edge. The $TKE$ and the $\omega$ profiles for the upper surface at the trailing edge can be written as $TKE(I_{TEu}, J)$ and $\omega(I_{TEu}, J)$ ($J = 1, J_{MAX}$), where $I$ is the index for the streamwise component of the body-fitted coordinate system and $J$ is the index for the normal-to-wall component. Here, $I_{TEu}$ corresponds to the $I$ index at the upper surface trailing edge location and $J_{MAX}$ stands for the $J$ index at the outer boundary of the grid in the normal direction. For the lower surface, the $TKE$ and the $\omega$ profiles can be written as $TKE(I_{TEl}, J)$ and $\omega(I_{TEl}, J)$ ($J = 1, J_{MAX}$), where, $I_{TEl}$ is the $I$ index at the lower surface trailing edge location. It should be noted that the maximum value of the $TKE$ is obtained within the boundary layer of the airfoil or the wing.
Figure A.1: The trailing edge plane of a wing grid used in three-dimensional parametric studies. The $TKE$ and $\omega$ values are extracted from the cell centers at the trailing edge plane.

In this region, $I = constant$ lines are perpendicular to the airfoil surface, since the grid lines are created by hyperbolic extrusion in the normal direction. Outside the boundary layer, these coordinate lines may become no longer normal to the wall, however the $TKE$ values in this region are approximately zero and do not affect the accuracy of finding the maximum value of the $TKE$.

- For three-dimensional (finite-wing problems), the same procedure described above must be repeated at each spanwise station for constructing the $TKE$ and the $\omega$ profiles. With a similar convention as used in two-dimensional studies, the $TKE$ and the $\omega$ profiles for the upper and the lower surface at the trailing edge along the span can be written as $TKE(I_{TEu,l}, J, K)$ and $\omega(I_{TEu,l}, J, K)$ ($J = 1, JMAX$ and $K = 1, KMAX$), where $K$ is the index for the spanwise component of the body-fitted coordinate system and $KMAX$ stands for the $K$ index at the wing tip. Figure A.1 depicts the geometric details of obtaining the $TKE$ and $\omega$ profiles at the trailing edge of a wing used in the parametric noise metric calculations.

The above procedure gives the $TKE$ and the $\omega$ values at the cell center locations, in other words, at discrete points. It should also be noted that the grid spacing in the
Figure A.2: The turbulent kinetic energy (TKE) profile at the trailing edge of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The black circles show the TKE values at the cell center locations and black arrow points the maximum value. The red line represents the 2nd order polynomial fit to the TKE in the vicinity of the maximum obtained from the cell center values and the red circle is the maximum value ($\tilde{TKE}_{max}$) obtained from the 2nd order polynomial fit.

The normal-to-wall direction is not uniform: fine grid spacing is required near the wall to resolve the laminar sublayer of the turbulent boundary layer. However, since such fine resolution is not required outside the boundary layer, the distance between each grid point in the normal direction is gradually increased starting from the wall region to improve computational efficiency. Therefore, seeking the maximum value of the TKE only at the cell centers and using the $\omega$ values at the corresponding locations create some problems, namely numerical noise, especially in the spanwise distribution of the characteristic length scale. In order to fix this, a special procedure is applied to both two- and three-dimensional CFD simulation results for extracting $u_0$ and $l_0$:

For $K = 1, KMAX$ (Note that $KMAX = 1$ for two-dimensional problems),

1. Calculate the distance from the wall, $z_n(I_{TE}, J, K)$ for each cell center location at
Figure A.3: The turbulence frequency ($\omega$) profile at the trailing edge of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The black circles show the $\omega$ values at the cell center locations and black arrow points the $\omega$ at the maximum TKE location obtained from the cell center values. The red line represents the $2^{nd}$ order polynomial fit to the $\omega$ and the red circle is the $\tilde{\omega}$ value obtained from the fit at the $\tilde{TKE}_{max}$ location.

the trailing edge ($J = 1, J_{MAX}$):

$$z_n(I_{TE}, J, K) = \sqrt{[x(I_{TE}, J, K) - x_w]^2 + [y(I_{TE}, J, K) - y_w]^2 + [z(I_{TE}, J, K) - z_w]^2},$$

where $x_w = x(I_{TE}, 1, K)$, $y_w = y(I_{TE}, 1, K)$, and $z_w = z(I_{TE}, 1, K)$. Here $I_{TE}$ index can stand for the upper or the lower surface.

2. Find the cell center where TKE is the maximum ($J = JM$).

3. Fit a $2^{nd}$ order polynomial to the TKE at $z_n(I_{TE}, JM - 1, K)$, $z_n(I_{TE}, JM, K)$, and $z_n(I_{TE}, JM + 1, K)$:

$$\tilde{TKE}(z_n) = az_n^2 + bz_n + c$$

The coefficients $a$, $b$, and $c$ are determined by solving three linear equations, which are obtained by evaluating $\tilde{TKE}(z_n)$ at the points $z_n(I_{TE}, JM - 1, K)$, $z_n(I_{TE}, JM, K)$, and $z_n(I_{TE}, JM + 1, K)$.
Figure A.4: The characteristic length scale at the trailing edge of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The black circles show the length scale values at the cell center locations and black arrow points the $l_0$ at the maximum $TKE$ location obtained from the cell center values. The red circle is the $l_0$ value obtained with the numerical procedure described in Appendix A.

4. Calculate the maximum value of the turbulent kinetic energy at the $K^{th}$ spanwise station, $\tilde{TKE}_{max}(K)$, and its location, $(\tilde{z}_n)_{max}(K)$ from the $2^{nd}$ order polynomial fit. Figure A.2 shows the $TKE$ profile at the trailing of an airfoil used in the noise metric studies. The maximum $TKE$ value obtained at the cell center location and $\tilde{TKE}_{max}$ value obtained from the $2^{nd}$ order polynomial fit can be seen in this figure.

5. Fit a $2^{nd}$ order polynomial to the $\omega$ at $z_n(I_{TE}, JM - 1, K)$, $z_n(I_{TE}, JM, K)$, and $z_n(I_{TE}, JM + 1, K)$ to obtain:

$$\tilde{\omega}(z_n) = dz_n^2 + ez_n + f$$

The coefficients $d, e, f$ are determined by solving three linear equations, which are obtained by writing $\tilde{\omega}(z_n)$ at the points $z_n(I_{TE}, JM - 1, K)$, $z_n(I_{TE}, JM, K)$, and $z_n(I_{TE}, JM + 1, K)$.

6. Obtain the turbulence frequency at the maximum turbulent kinetic energy loca-
Figure A.5: A portion of the length scale ($l_0$) distribution along the span of a wing used in the preliminary noise metric studies. The symbols show the length scale distribution obtained before and after the application of the numerical procedure described in Section A.

7. Finally calculate the characteristic turbulent velocity and the turbulence length scale at the $K^{th}$ spanwise station as:

\[ u_0(K) = \sqrt{TKE_{\text{max}}(K)} \quad \text{and} \quad l_0(K) = \frac{u_0(K)}{\bar{\omega}(K)}. \]

Figure A.4 shows the characteristic length scale profile at the trailing of an airfoil (SC2-0714 airfoil, $C_l = 0.788$) used in the noise metric studies. The location and the magnitude of the length scale value obtained with the polynomial-fit procedure can be seen from this Figure.

The success of the numerical procedure described above can be seen from Figure A.5. This figure shows the length scale $l_0$ distribution along the span of a wing used in the preliminary noise metric studies. The numerical noise can be noticed for the case where
the cell center values of maximum $TKE$ and $\omega$ are used to calculate the length scale at each spanwise station. When the polynomial fit procedures are applied to the data to find the maximum $TKE$ and the $\omega$ values, the improvement in the smoothness of the length scale distribution can be seen.
Appendix B

Remarks On CFD Simulation Uncertainties

B.1 Introduction

Computational fluid dynamics (CFD) has become an important aero/hydrodynamic analysis and design tool in recent years. CFD simulations with different levels of fidelity, ranging from linear potential flow solvers to full Navier-Stokes codes, are widely used in the multidisciplinary design and optimization (MDO) of advanced aerospace and ocean vehicles.\textsuperscript{66} Although low-fidelity CFD tools have low computational cost and are easily used, the full viscous equations are needed for the simulation of complex turbulent separated flows, which occur in many practical cases such as high-angle-of attack aircraft, high-lift devices, maneuvering submarines and missiles.\textsuperscript{67} Even for cases when there is no flow separation, the use of high-fidelity CFD simulations is desirable for obtaining higher accuracy. Due to modelling, discretization, and computation errors, the results obtained from CFD simulations have a certain level of uncertainty. It is important to understand the sources of CFD simulation errors and their magnitudes to be able to assess the magnitude of the uncertainty in the results.

The objective of this work is to illustrate different sources of uncertainty in CFD simulations, by a careful study of a typical, but complex fluid dynamics problem. We will try to compare the magnitude and importance of each source of uncertainty.

The test problem in this study is a two-dimensional, turbulent, transonic flow in a
converging-diverging channel. CFD calculations are done with the General Aerodynamic Simulation Program (GASP).\textsuperscript{43} Runs were performed with different turbulence models, grid densities, and flux-limiters to see the effect of each on the CFD simulation uncertainties. In addition to these, the contribution of the error in geometry representation to the CFD simulation uncertainties is studied through the use of a modified geometry, based on the measured geometric data. The exit station of the diffuser and the exit pressure ratio are varied to determine the effects of changes of the downstream boundary conditions on the results. This study provides detailed information about the sources and magnitudes of uncertainties associated with the numerical simulation of flow fields that have strong shocks and shock-induced separated flows.

B.2 Uncertainty Sources

To better understand the accuracy of CFD simulations, the main sources of errors and uncertainties should be identified. Oberkampf and Blottner\textsuperscript{68} classified CFD error sources. In their classification, the error sources are grouped under four main categories: (1) physical modelling errors, (2) discretization and solution errors, (3) programming errors, and (4) computer round-off errors.

Physical modelling errors originate from the inaccuracies in the mathematical models of the physics. The errors in the partial differential equations (PDEs) describing the flow, the auxiliary (closure) physical models, and the boundary conditions for all the PDEs are included in this category. Turbulence models used in viscous calculations are considered as one of the auxiliary physical models, usually the most important one. They are used for modelling the additional terms that originate as the result of Reynolds averaging, which in itself is a physical model.

Oberkampf and Blottner\textsuperscript{68} define discretization errors as the errors caused by the numerical replacement of PDEs, the auxiliary physical models and continuum boundary conditions by algebraic equations. Consistency and the stability of the discretized PDEs, spatial (grid) and temporal resolution, errors originating from the discretization of the continuum boundary conditions are listed under this category. The difference between the exact solution to the discrete equations and the approximate (or computer) solution is defined as the solution error of the discrete equations. Iterative convergence error of
the steady-state or the transient flow simulations is included in this category. A similar
description of the discretization errors can also be found in Roache,\textsuperscript{69,70} and Pelletier \textit{et al.}\textsuperscript{71}

Since the terms \textit{error} and \textit{uncertainty} are commonly used interchangeably in many CFD
studies, it will be useful to give a definition for each. Uncertainty, itself, can be defined
in many forms depending on the application field as listed in DeLaurentis and Mavris.\textsuperscript{72}
For computational simulations, Oberkampf \textit{et al.}\textsuperscript{73,74} described \textit{uncertainty} as a poten-
tial deficiency in any phase or activity of modelling process that is due to the lack of
knowledge, whereas \textit{error} is defined as a recognizable deficiency in any phase or activity
of modelling and simulation.

Considering these definitions, any deficiency in the physical modelling of the CFD activ-
ities can be regarded as \textit{uncertainty} (such as uncertainty in the accuracy of turbulence
models, uncertainty in the geometry, uncertainty in thermophysical parameters \textit{etc.}),
whereas the deficiency associated with the discretization process can be classified as
\textit{error}.\textsuperscript{74}

Discretization errors can be quantified by using methods like Richardson’s extrapolation
or grid-convergence index (GCI), a method developed by Roache\textsuperscript{70} for uniform reporting
of grid-convergence studies. However, these methods require fine grid resolution in the
asymptotic range, which may be hard to achieve in the simulation of flow fields around
complex geometries. Also, non-monotonic grid convergence, which may be observed in
many flow simulations, prohibits or reduces the applicability of such methods. That
is, it is often difficult to estimate errors in order to separate them from uncertainties.
Therefore, for the rest of the paper, the term \textit{uncertainty} will be used to describe the
inaccuracy in the CFD solution variables originating from discretization, solution, or
physical modelling errors.

\section*{B.3 Transonic Diffuser Case}

\subsection*{B.3.1 Description of the physical problem}

The test case presented in this paper is the simulation of a 2-D, turbulent, transonic
flow in a converging-diverging channel, known as the \textit{Sajben Transonic Diffuser} in CFD
The exit station is at \( x/h_t = 8.65 \) for the geometry shown at the top part of Figure B.1, where \( h_t \) is the throat height. This is the original geometry used in the computations and a large portion of the results with different solution and physical modelling parameters are obtained with this version. The exit station is located at \( x/h_t = 14.44 \) for the other geometry shown in Figure B.1. This extended geometry is used to study the effect of varying the downstream boundary location on the CFD simulation results. For both geometries, the bottom wall of the channel is flat and the converging-diverging section of the top wall is described by an analytical function of \( x/h_t \) defined in Bogar et al. In addition to these two geometries, a third version of the same diffuser (the modified-wall geometry) has been developed for this research and has been used in our calculations. This version has the same inlet and exit locations as the original geometry, but the upper wall is described by natural cubic splines fitted to the geometric data points that were measured in the experimental studies. Having observed the fact that the upper wall contour obtained by the analytical equation and the contour described by experimental data points are slightly different, the modified-wall geometry is used to find the effects of geometric uncertainty on the numerical results.

Despite the relatively simple geometry, the flow has a complex structure. The exit pressure ratio \( P_e/P_{0i} \) sets the strength and the location of a shock that appears downstream of the throat (Figure B.2). In our studies, for the original and the modified-wall geometries, we define \( P_e/P_{0i} = 0.72 \) as the strong shock case and \( P_e/P_{0i} = 0.82 \) as the weak shock case. A separated flow region exists just after the shock at \( P_e/P_{0i} = 0.72 \). Although a nominal exit station was defined at \( x/h_t = 8.65 \) for the diffuser used in the experiments, the physical exit station is located at \( x/h_t = 14.44 \). In the experiments, \( P_e/P_{0i} \) was measured as 0.7468 and 0.8368 for the strong and the weak shock cases respectively at the physical exit location. Table B.1 gives a summary of the different versions of the transonic diffuser geometry and exit pressure ratios used in the computations.

A large set of experimental data for a range of exit pressure ratios are available. In our study, top and bottom wall pressure values were used for the comparison of CFD results with the experiment. Note that the diffuser geometry used in the experiments has suction slots placed at \( x/h_t = 9.8 \) on the bottom and the side walls to limit the growth of the boundary layer. The existence of these slots can affect the accuracy of the quantitative comparison between the experiment and the computation at the downstream locations.
B.3.2 Computational modelling

CFD calculations are performed with GASP, a Reynolds-averaged, three-dimensional, finite-volume, Navier-Stokes code, which is capable of solving steady-state (time asymptotic) and time-dependent problems. For this problem, the inviscid fluxes were calculated by an upwind-biased third-order spatially accurate Roe flux scheme. The minimum modulus (Min-Mod) and Van Albada’s flux limiters were used to prevent non-physical oscillations in the solution. All the viscous terms were included in the solution and two turbulence models, Spalart-Allmaras\textsuperscript{77} (Sp-Al) and $k$-$\omega$\textsuperscript{57} (Wilcox, 1998 version) with Sarkar’s Compressibility Correction, were used for modelling the viscous terms.

The adiabatic no-slip boundary condition was used on the top and the bottom walls of the transonic diffuser geometry. At the inlet, a constant total pressure ($P_{0i}$) and temperature ($T_{0i}$) were specified (subsonic $P_{0i}$-$T_{0i}$ inflow boundary condition in GASP). The static pressure was taken from the adjacent interior cell and the other flow variables were calculated by using isentropic relations. At the exit, the outflow boundary was set to a constant static pressure ($P_e$), while the remaining flow variables were extrapolated from the interior cells. To initialize each CFD solution, inflow conditions were used.

The iterative convergence of each solution is examined by monitoring the overall residual, which is the sum (over all the cells in the computational domain) of the $L^2$ norm of all the governing equations solved in each cell. In addition to this overall residual information, the individual residual of each equation and some of the output quantities are also monitored.

The sizes and the nomenclature of the grids used in the computations are given in Table B.2. Grid 2 (top) and Grid 2\textsubscript{ext} (bottom) are shown in Figure B.1. To resolve the flow gradients due to viscosity, the grid points were clustered in the $y$-direction near the top and the bottom walls. In wall bounded turbulent flows, it is important to have a sufficient number of grid points in the wall region, especially in the laminar sublayer, for the resolution of the near wall velocity profile, when turbulence models without wall-functions are used. A measure of grid spacing near the wall can be obtained by examining the $y^+$ values defined as

$$y^+ = \frac{y\sqrt{\tau_w/\rho}}{\nu},$$

where $y$ is the distance from the wall, $\tau_w$ the wall shear stress, $\rho$ the density of the fluid,
and \( \nu \) the kinematic viscosity. In turbulent boundary layers, a \( y^+ \) value between 7 and 10 is considered as the edge of the laminar sublayer. General CFD practice has been to have several mesh points in the laminar sublayer with the first mesh point at \( y^+ = O(1) \). In our study, the maximum value of \( y^+ \) values for Grid 2 and Grid 3 at the first cell center locations from the bottom wall were found to be 0.53 and 0.26 respectively. The grid points were also stretched in the \( x \)-direction to increase the grid resolution in the vicinity of the shock wave. The center of the clustering in the \( x \)-direction was located at \( x/h_t = 2.24 \). At each grid level, except the first one, the initial solution estimates were obtained by interpolating the primitive variable values of the previous grid solution to the new cell locations. This method, known as *grid sequencing*, was used to reduce the number of iterations required to converge to a steady state solution at finer mesh levels.

It should be noted that grid levels such as \( g5, g4, \) and \( g4_{ext} \) are more highly refined than those normally used for typical two-dimensional problems and well beyond what could be used in a three-dimensional flow simulation. A single solution on Grid 5 required approximately 1170 hours of total node CPU time on a SGI Origin2000 with six processors, when 10000 cycles were run with this grid. If we consider a three-dimensional case, with the addition of another dimension to the problem, Grid 2 would usually be regarded as a fine grid, whereas Grid 3, 4, and 5 would generally not be used.

### B.4 Results and Discussion

For the transonic flow in the converging-diverging channel, the uncertainty of the CFD simulations is investigated by examining the nozzle efficiency \( (n_{eff}) \) as a global output quantity obtained at different \( P_e/P_{0i} \) ratios with different grids, flux limiters (Min-Mod and Van Albada), and turbulence models (Sp-Al and \( k-\omega \)). The nozzle efficiency is defined as

\[
    n_{eff} = \frac{H_{0i} - H_e}{H_{0i} - H_{es}} ,
\]

where \( H_{0i} \) is total enthalpy at the inlet, \( H_e \) the enthalpy at the exit, and \( H_{es} \) the exit enthalpy at the state that would be reached by isentropic expansion to the actual pressure at the exit. Since the enthalpy distribution at the exit was not uniform, \( H_e \) and \( H_{es} \) were obtained by integrating the cell-averaged enthalpy values across the exit plane. Besides \( n_{eff} \), wall pressure values from the CFD simulations are compared with experimental...
data.

In the transonic diffuser study, the uncertainty in CFD simulation results has been studied in terms of five contributions: (1) iterative convergence error, (2) discretization error, (3) error in geometry representation, (4) turbulence model, and (5) changing the downstream boundary condition. In particular, (1) and (2) contribute to the numerical uncertainty, which is the subject of the verification process; (3), (4), and (5) contribute to the physical modelling uncertainty, which is the concern of the validation process. The main observations on the sources of uncertainties are summarized in Table B.3.

### B.4.1 The Iterative convergence error

The convergence of each transonic diffuser case to a steady-state solution has been examined by using various $L^2$ norm residuals and the $n_{eff}$ results. The overall residual and the residual of each equation were monitored at every iteration, whereas the $n_{eff}$ results were checked at certain iteration numbers. Figure B.3 shows the convergence history of the $L^2$ norm residual of the energy equation for the strong shock case obtained with the Sp-Al turbulence model and the original geometry. The convergence history of the residual, normalized by its initial value, is presented for both limiters and the grid levels g1, g2, g3, and g4. By examining this figure, it can be seen that the main parameter that affects the residual convergence of a solution is the flux-limiter. With the Min-Mod limiter, the residuals of Grid g2, g3, and g4 do not reach even one order of magnitude reduction while the same grid levels show much better residual convergence when the Van Albada limiter is used. For example, the residual of Grid 3 was reduced more than seven orders of magnitude when 10000 cycles were run with the Van Albada limiter. The same convergence behavior of the Min-Mod and the Van Albada limiter was observed for the residual of the other equations and the weak shock case. The $k$-$\omega$ turbulence model also exhibited the same convergence behavior for Min-Mod and Van Albada limiters at both shock conditions.

Although the use of the Min-Mod limiter causes poor $L^2$ norm residual convergence, this does not seem to affect the final results, such as the wall pressure values or the nozzle efficiencies. Figure B.4 shows the convergence history of nozzle efficiency at different grid levels for the Sp-Al, Min-Mod, strong shock case obtained with the original geometry.
The convergence can be seen qualitatively at all grid levels for this scale of $n_{eff}$ axis. However, at a smaller scale, small oscillations have been observed in nozzle efficiency results of Grid g4 and g5 starting from iteration number 10000. The amplitude of the oscillations (the fluctuating component of the $n_{eff}$) were on the order of $10^{-4}$ after the iteration number 13000 for Grid g5. As will be seen in the next section, the magnitude of the discretization error is much higher compared to the order of the iterative convergence error, especially in the coarser grid levels.

Although a steady-state solution is sought for each case, the physical problem itself may have some unsteady characteristics, such as the oscillation of the shock wave, which is a common phenomena observed in the transonic flows. Hsieh and Coakley\textsuperscript{78} studied the unsteady nature of the shock in the Sajben Diffuser geometry by changing the exit location. They used a physical time step of $2.77 \times 10^{-6}$ seconds to resolve the time-dependent shock oscillations and wall pressures. In this study, time-dependent runs were performed with grid levels g2 and g3 by using a physical time step of $10^{-2}$ seconds and no change in nozzle efficiency values, thus no unsteady effects were observed at that time-scale. In this study, it may be more appropriate to consider the output variables such as the $n_{eff}$ values or the wall pressures obtained from the steady-state CFD runs as the mean time-averaged values of the corresponding quantities over a longer time-scale than the one used in typical Reynolds time-averaging.

\section*{B.4.2 The discretization error}

In order to investigate the contribution of the discretization error to the uncertainty in CFD simulation results, we study the Sp-Al and $k$-$\omega$ cases separately. Grid level and flux-limiter affect the magnitude of the discretization error. Grid level determines the spatial resolution, and the limiter is part of the discretization scheme, which reduces the spatial accuracy of the method to first order in the vicinity of shock waves.

A qualitative assessment of the discretization error in nozzle efficiency results obtained with the original geometry can be made by examining Figure B.5. The largest value of the difference between the strong shock results of Grid 2 and Grid 4 is observed for the case with Sp-Al model and the Min-Mod limiter. For the weak shock case, the difference between each grid level is not as large as that of the strong shock case when the results obtained with the Sp-Al turbulence model are compared. Weak shock results
in Figure B.5 also show that the \( k-\omega \) turbulence model is slightly better than the Sp-Al in terms of the discretization error for this pressure ratio. Non-monotonic behavior of the \( k-\omega \) results can be seen for the strong shock case as the mesh is refined, whereas the same turbulence model shows monotonic convergence for the weak shock cases. The Sp-Al turbulence model exhibits monotonic convergence in both shock conditions.

Richardson’s extrapolation technique has been used to estimate the magnitude of the discretization error at each grid level for cases that show monotonic convergence. This method is based on the assumption that \( f_k \), a local or global output variable obtained at grid level \( k \), can be represented by

\[
f_k = f_{\text{exact}} + \alpha h^p + O(h^{p+1}), \tag{B.3}
\]

where \( h \) is a measure of grid spacing, \( p \) the order of the method, and \( \alpha \) the \( p \)th-order error coefficient. Note that Equation B.3 will be valid when \( f \) is smooth and in the asymptotic grid convergence range. In most cases, the observed order of spatial accuracy is different than the nominal (theoretical) order of the numerical method due to factors such as the existence of discontinuities in the solution domain, boundary condition implementation, flux-limiters, etc. Therefore, the observed value of \( p \) should be determined and used in the calculations required for approximating \( f_{\text{exact}} \) and the discretization error. Calculation of the approximate value of the observed order of accuracy (\( \tilde{p} \)) needs the solutions from three grid levels, and the estimate of the \( f_{\text{exact}} \) value requires two grid levels. The details of the calculation procedure is given in Hosder et al.\(^{79}\)

Table B.4 summarizes the discretization error in \( n_{\text{eff}} \) results obtained with the original geometry. When the results at grid level \( g2 \) are compared, the Sp-Al, Min-Mod, and \( P_e/P_{bi} = 0.72 \) case has the highest discretization error (6.97%), while the smallest error (1.45%) is obtained with \( k-\omega \) turbulence model at \( P_e/P_{bi} = 0.82 \). The finest grid level, \( g5 \) was used only for the Sp-Al, Min-Mod, strong shock case obtained with the original geometry.

In Table B.4, the observed order of accuracy \( \tilde{p} \), is smaller than the nominal order of the scheme and its value is different for each case with a different turbulence model, limiter, and shock condition. The values of both \( (\tilde{n}_{\text{eff}})_{\text{exact}} \) and \( \tilde{p} \) also depend on the grid levels used in their approximations. For example, the \( \tilde{p} \) value was calculated as 1.322 and 1.849 for the Sp-Al, Min-Mod, strong shock case with different grid levels. These nonintegral
\( \tilde{p} \) values indicate grid convergence has not yet occurred, so the approximation of the discretization error at each grid level by Richardson’s extrapolation is inaccurate (the source of some uncertainty).

The difference in nozzle efficiency values due to the choice of the limiter can be seen in the results of Grid 1 and Grid 2 for the strong shock case and Grid 1 for the weak shock case. The maximum difference between the Min-Mod limiter and Van Albada limiter occurs on Grid 1 with the Sp-Al model. The relative uncertainty due to the choice of the limiter is more significant for the strong shock case. For both pressure ratios, the solutions obtained with different limiters give approximately the same nozzle efficiency values as the mesh is refined.

Figure B.6 shows the significance of the discretization uncertainty between each grid level. In this figure, the noisy behavior of \( n_{\text{eff}} \) results obtained with Grid 1 can be seen for both turbulence models. The order of the noise error is much smaller than the discretization error between each grid level, however this can be a significant source of uncertainty if the results of Grid 1 are used in a gradient based optimization.

When we look at Mach number values at two points in the original geometry, one, upstream of the shock \( (x/h_{\text{t}} = -1.5) \) and the other, downstream of the shock \( (x/h_{\text{t}} = 8.65, \) the exit plane), both of which are located at the mid point of the local channel heights (Figure B.7), we see the convergence of Mach number upstream of the shock for all the cases. However, for the strong shock case, the lack of convergence downstream of the shock at all grid levels with the \( k-\omega \) model can be observed. For the Sp-Al case, we see the convergence only at grid levels g3 and g4. For the weak shock case, downstream of the shock, the convergence at all grid levels with the \( k-\omega \) model is also seen. At this pressure ratio, Sp-Al model results do not seem to converge, although the difference between each grid level is small. These results may again indicate the effect of the complex flow structure downstream of the shock, especially the separated flow region seen in the strong shock case, on the grid convergence.

### B.4.3 Error in the geometry representation

The contribution of the error in geometry representation to CFD simulation uncertainties is studied by comparing the results of the modified-wall and the original geometry
obtained with the same turbulence model, limiter, and grid level. Figure B.8 gives the percent error distribution in $y/h_t$ (difference from the analytical value) for the upper wall of the modified-wall geometry at the data points measured in the experiments. Natural cubic splines are fit to these data points to obtain the upper wall contour. The difference between the upper wall contours of the original and the modified-wall geometry in the vicinity of the throat location is shown in Figure B.9.

The flow becomes supersonic just after the throat and is very sensitive to the geometric irregularities for both $P_e/P_{oi} = 0.72$ and 0.82. From the top wall pressure distributions shown in Figures B.10 and B.11, a local expansion/compression region can be seen around $x/h_t = 0.5$ with the modified-wall geometry. This is due to the local bumps created by two experimental data points, the third and the fifth ones from the throat (Figure B.9).

Since neither the wall pressure results obtained with the original geometry nor the experimental values have this local expansion/compression, the values of these problematic points may contain some measurement error. The locations of these two points were modified by moving them in the negative $y$-direction halfway between their original value and the analytical equation value obtained at the corresponding $x/h_t$ locations. These modified locations are shown with black circles in Figure B.9. The wall pressure results of the geometry with the modified experimental points (Figures B.10 and B.11) show that the local expansion/compression region seems to be smoothed, although not totally removed. One important observation that can be made from the same figures is the improvement of the match between the CFD results and the experiment upstream of the throat when the modified-wall geometry is used. Since the viscous effects are important only in a very thin boundary layer near the wall region where there is no flow separation, contribution of the Sp-Al or the $k$-$\omega$ turbulence models to the overall uncertainty is very small upstream of the shock for both $P_e/P_{oi} = 0.72$ and 0.82.

### B.4.4 Evaluation with the orthogonal distance error

The quantitative comparison of CFD simulation results with the experiment can be done considering different measures of error. In the transonic diffuser case, we use the orthogonal distance error $E_o$ to approximate the difference between the wall pressure values obtained from the numerical simulations and the experimental data. The error
$E_n$ is defined as

$$E_n = \frac{1}{N_{\text{exp}}} \sum_{i=1}^{N_{\text{exp}}} d_i,$$

where

$$d_i = \min_{x_{\text{inlet}} \leq x \leq x_{\text{exit}}} \left[ \left( x - x_i \right)^2 + \left( P_c(x) - P_{\text{exp}}(x_i) \right)^2 \right]^{1/2}.$$

In equations (B.4) and (B.5), $d_i$ represents the orthogonal distance between the $i^{th}$ experimental point and the $P_c(x)$ curve (the wall pressure obtained from the CFD calculations), $P_{\text{exp}}$ is the experimental wall pressure value, and $N_{\text{exp}}$ is the number of experimental data points used. Pressure values are scaled by $P_{0i}$ and the $x$ values are scaled by the length of the channel.

The error $E_n$ was evaluated separately in two regions: upstream of the experimental shock location (UESL) and downstream of the experimental shock location (DESL). The details of $E_n$ calculations can be found in Hosder et al.$^{79}$ Table B.5 lists the top wall scaled error $\hat{E}_n$ values obtained for UESL with the original geometry, different grids, turbulence models, and flux-limiters. Table B.6 gives DESL results.

It can be seen from Table B.5 that the results obtained with the Sp-Al and the $k$-$\omega$ turbulence models are very close, especially for the weak shock case, when the values at the grid level g4 are compared. For each $P_e/P_{0i}$, the small difference between the results of each turbulence model at the finest mesh level originate from the difference in the shock locations obtained from the CFD calculations. This again shows that a large fraction of the uncertainty observed upstream of the shock (UESL) in the wall pressure values originates from the uncertainty in the geometry representation. The difference in $\hat{E}_n$ between each grid level for each turbulence model and $P_e/P_{0i}$ is very small indicating that the wall pressure distributions upstream of the shock obtained at each grid level are approximately the same. In other words, grid convergence is achieved upstream of the shock and the discretization error in wall pressure values at each grid level is very small.

Recall that the experimental data also contains uncertainty originating from many factors such as geometric irregularities, difference between the actual $P_e/P_{0i}$ and its intended value, measurement errors, heat transfer to the fluid, etc. We have discussed the error due to geometric irregularities in the previous section. In a way, this error in geometry representation can also be regarded as a part of the uncertainty in the experimental data. By evaluating the orthogonal distance error in two separate regions, DESL and UESL,
we tried to approximate the contribution of the geometric uncertainty to the CFD results obtained with the original geometry. However, experimental wall pressure values may still have a certain level of uncertainty associated with the remaining factors.

B.4.5 Turbulence models

To approximate the contribution of the turbulence models to the CFD simulation uncertainties in the transonic diffuser case, $\hat{E}_n$ values calculated for the top wall pressure distributions downstream of the shock (DESL) (Table B.6) at grid level g4 are examined. By considering the results of the finest mesh level, the contribution of the discretization error should be minimized, although it is difficult to isolate the numerical errors completely from the physical modelling uncertainties, especially for the strong shock case.

The Sp-Al turbulence model is more accurate than the $k$-$\omega$ model for the strong shock case. In fact, the difference is significant, with $k$-$\omega$ giving the highest error of all the cases, which is larger than the Sp-Al error by a factor of 3.6. With the Sp-Al model, the orthogonal distance error gets smaller as the mesh is refined, while the $k$-$\omega$ model gives the largest error value at grid level g4. When compared to the error values presented in Table B.5, for the strong shock, the uncertainty of the $k$-$\omega$ turbulence model is 3.7 times larger than the error due to the geometric uncertainty. On the other hand, the uncertainty of the Sp-Al model has approximately the same magnitude as the geometric uncertainty.

As opposed to the strong shock case, the $k$-$\omega$ turbulence model gives more accurate wall pressure distributions than the Sp-Al model when the weak shock results of grid g4 are compared (Table B.6). The orthogonal distance error of Sp-Al is twice as big as that of the $k$-$\omega$ model. The minimum error for the Sp-Al model is obtained at grid level g2, while the wall pressure distributions of the $k$-$\omega$ model get closer to the experimental distribution as the mesh is refined. The results of the Sp-Al model show that the most accurate results are not always obtained at the finest mesh level. The error due to the geometric uncertainty is bigger than the uncertainty of the $k$-$\omega$ model by a factor of 2.6 in the weak shock case. The uncertainty of the Sp-Al model is slightly smaller than the geometric uncertainty for the same shock condition.
B.4.6 Downstream boundary condition

The effect of the downstream boundary location variation on the CFD simulation results of the transonic diffuser case has been investigated by using the extended geometry, which has the physical exit station at the same location as the geometry used in the actual experiments. For the strong shock case, the runs were performed with the Sp-Al model and two $P_e/P_{0i}$ ratios, 0.72 and 0.7468. The second pressure ratio is the same value measured at the physical exit station of the geometry used in the experiments for the strong shock case. The results obtained with the extended geometry were compared to the results of the original geometry.

Figure B.12 shows the streamline patterns of the separated flow region obtained with different geometries and $P_e/P_{0i}$ ratios in the strong shock case. The comparison of the separation bubble size is given in Figure B.13. The separation bubble obtained with the extended geometry and $P_e/P_{0i} = 0.72$ is bigger and extends farther in the downstream direction compared to the other two cases. The separation bubbles obtained with the original geometry ($P_e/P_{0i} = 0.72$) and the extended geometry ($P_e/P_{0i} = 0.7468$) are approximately the same in size. These results are also consistent with the top wall pressure distributions given in Figure B.14.

With the extended geometry and $P_e/P_{0i} = 0.72$, the flow accelerates more under the separation bubble, and the pressure is lower compared to the other cases where the separation bubbles have smaller thickness. Moving the exit location further downstream increases the strength of the shock and the size of the separation region. As the shock gets stronger, its location is shifted downstream. On the other hand, increasing $P_e/P_{0i}$ reduces the strength of the shock, and moves the shock location upstream.

B.4.7 Discussion of uncertainty on nozzle efficiency

We use nozzle efficiency as a global indicator of the CFD results in the transonic diffuser case and the scatter in the computed values of this quantity originates from the use of different grid levels, limiters, turbulence models, geometries, and boundary conditions for each shock strength case. A graphical representation of this variation is given in Figure B.5. This figure shows a cloud of results that a reasonably informed user may obtain from CFD calculations. The numerical value of each point is presented in Table B.7. We
will analyze the scatter in nozzle efficiency results starting from grid level 2, since the coarse Grid 1 will not be used by those that have significant experience in performing CFD simulations. On the other hand, grid levels 3 and 4 would generally not be used in practical CFD applications, particularly in three dimensions, due to their computational expense.

For the purpose of determining the variation in nozzle efficiency in terms of a percent value, we use the g4, Sp-Al, Van-Albada result as the comparator. When we consider the cases obtained with the original geometry, maximum variation for the strong shock condition is 9.9% and observed between the results of g2, $k-\omega$, Min-Mod and g4, Sp-Al, and Van Albada. Maximum difference in the weak shock results is 3.8% and obtained between the results of g2, $k-\omega$, Van Albada and g4, Sp-Al, and Min-Mod.

For each case with a different turbulence model and limiter, the variation between the results of g2 and g4 may be used to get an estimate of the uncertainty due to discretization error. The maximum variation for the strong shock is 5.7% and obtained with Sp-Al model and the Min-Mod limiter. For the weak shock case, the maximum difference is 3.5% and obtained with the same turbulence model and limiter.

We can approximate the relative uncertainty originating from the selection of different turbulence models by comparing the nozzle efficiency values obtained with the same limiter and the grid level. At grid level 4, the maximum difference between the strong shock results of Sp-Al and $k-\omega$ model is 9.2% and obtained with the Min-Mod limiter. For the weak shock case, the maximum difference at grid level 2 is 2.2%, and obtained with the same limiter. It should be noted that, at each grid level, relative uncertainty due to the turbulence models is different resulting from the interaction of physical modelling uncertainties with the numerical errors.

For the strong shock case, at each grid level, the difference between nozzle efficiency values of the original geometry and the results of the modified-wall geometry is much smaller than the variations originating from the other sources of uncertainty regardless of the turbulence model and the limiter used. On the other hand, this difference is notable for the weak shock case and varies between 0.9% and 1.4%.

Nozzle efficiency values of the extended geometry show considerable deviation from the results of the original geometry at certain grid levels, when 0.7468 and 0.8368 are used as the exit pressure ratios for the strong and the weak shock cases, respectively. For the exit
pressure ratio of 0.7468, the maximum difference is 1.8% and obtained with grid level 3. The maximum difference for the exit pressure ratio of 0.8368 is 6.9% and observed at grid level 4. The difference between the results of the original and the extended geometry is smaller when the exit pressure ratios of 0.72 and 0.82 are used. For the exit pressure ratio of 0.72, the maximum difference is 0.8% and observed at grid level 3. A maximum difference of 1.1% is obtained at grid level 2 for the exit pressure ratio of 0.82.

Main observations on the uncertainties in nozzle efficiencies are summarized in Table B.8.

**B.5 Conclusions**

Different sources of uncertainty in CFD simulations are illustrated by examining a 2-D, turbulent, transonic flow in a converging-diverging channel at various $P_e/P_{i0}$ ratios by using the commercial CFD code GASP. Runs were performed with different turbulence models (Sp-Al and $k$-$\omega$), grid levels, and flux-limiters (Min-Mod and Van Albada). Two flow conditions were studied by changing the exit pressure ratio: the first one was a complex case with a strong shock and a separated flow region; the second was a weak shock case with attached flow throughout the entire channel. The uncertainty in the CFD simulation results was studied in terms of five contributions: (1) iterative convergence error, (2) discretization error, (3) error in geometry representation, (4) turbulence model, and (5) downstream boundary condition. In addition to the original transonic diffuser geometry used in the calculations, the contribution of the error in geometry representation to the CFD simulation uncertainties was studied through the use of a modified geometry, based on the measured geometric data. Also an extended version of the transonic diffuser was used to determine the effect of a change of the downstream boundary location on the results.

Overall, this paper demonstrated that for the simulation of attached flows, informed CFD users can obtain reasonably accurate results, whereas they are more likely to get large errors for the cases that have strong shocks with substantial separation.

Both internal and external flow examples show that grid convergence is not achieved with grid levels that have moderate mesh sizes. Shocks and shock-induced separations have significant effect on the grid convergence. The magnitudes of discretization errors were influenced by the physical (turbulence) models used.
In some cases, turbulence modelling uncertainties and discretizations errors may cancel each other, and the closest results to the experiment can be obtained at intermediate grid levels. This shows the strong interaction among different types of uncertainties.

In nozzle efficiency results, the range of variation for the strong shock case was much larger than that observed in the weak shock case. The discretization errors were up to 6% and the relative uncertainty originating from the selection of different turbulence models was as large as 9% for the strong shock case. For the weak shock case, nozzle efficiency values were more sensitive to the exit boundary conditions and associated error magnitudes were larger than those from other sources. The difference between the results from the original geometry and the extended geometry was as large as 7% when the exit pressure ratio of 0.8368 was used. The contribution of the error in geometry representation to the overall uncertainty in nozzle efficiency results was up to 1.5% for the weak shock case, whereas this contribution was negligible for the strong shock case.

Overall, this study provides observations on CFD simulation uncertainties that may help the development of sophisticated methods required for the characterization and quantification of uncertainties associated with the numerical simulation of turbulent separated flows.
### B.6 Tables of Appendix B

Table B.1: Different versions of the transonic diffuser geometry and exit pressure ratios ($P_e/P_{0i}$) used in the computations.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>$x/h_i$ at the exit station</th>
<th>$P_e/P_{0i}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>original</td>
<td>8.65</td>
<td>0.72 and 0.82</td>
</tr>
<tr>
<td>modified-wall</td>
<td>8.65</td>
<td>0.72 and 0.82</td>
</tr>
<tr>
<td>extended</td>
<td>14.44</td>
<td>0.72, 0.7468</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.82, and 0.8368</td>
</tr>
</tbody>
</table>
Table B.2: Mesh size nomenclature for the transonic diffuser case. In the simulations, five different grids were used for the original geometry: Grid 1 (g1), Grid 2 (g2), Grid 3 (g3), Grid 4 (g4), and Grid 5 (g5). The finest mesh is Grid 5 and the other grids are obtained by reducing the number of divisions by a factor of 2 in both $x$- and $y$-directions at each consecutive level (grid halving). Grid 5 is used only for the case with the Sp-Al turbulence model, Min-Mod limiter, and $P_e/P_{0i} = 0.72$. Four grid levels were used for the extended geometry: Grid $1_{ext}$ ($g1_{ext}$), Grid $2_{ext}$ ($g2_{ext}$), Grid $3_{ext}$ ($g3_{ext}$), and Grid $4_{ext}$ ($g4_{ext}$). The grids for the extended geometry and the grids generated for the original geometry are essentially the same between the inlet station and $x/h_t = 8.65$. For the modified-wall geometry, three grid levels were used: Grid $1_{mw}$ ($g1_{mw}$), Grid $2_{mw}$ ($g2_{mw}$), and Grid $3_{mw}$ ($g3_{mw}$). All the grids have the same mesh distribution in the $y$-direction.

<table>
<thead>
<tr>
<th>Grid</th>
<th>$x/h_t$ at the exit station</th>
<th>mesh size</th>
</tr>
</thead>
<tbody>
<tr>
<td>g1</td>
<td>8.65</td>
<td>$41 \times 26 \times 2$</td>
</tr>
<tr>
<td>g2</td>
<td>8.65</td>
<td>$81 \times 51 \times 2$</td>
</tr>
<tr>
<td>g3</td>
<td>8.65</td>
<td>$161 \times 101 \times 2$</td>
</tr>
<tr>
<td>g4</td>
<td>8.65</td>
<td>$321 \times 201 \times 2$</td>
</tr>
<tr>
<td>g5</td>
<td>8.65</td>
<td>$641 \times 401 \times 2$</td>
</tr>
<tr>
<td>g1_{ext}</td>
<td>14.44</td>
<td>$46 \times 26 \times 2$</td>
</tr>
<tr>
<td>g2_{ext}</td>
<td>14.44</td>
<td>$91 \times 51 \times 2$</td>
</tr>
<tr>
<td>g3_{ext}</td>
<td>14.44</td>
<td>$181 \times 101 \times 2$</td>
</tr>
<tr>
<td>g4_{ext}</td>
<td>14.44</td>
<td>$361 \times 201 \times 2$</td>
</tr>
<tr>
<td>g1_{mw}</td>
<td>8.65</td>
<td>$41 \times 26 \times 2$</td>
</tr>
<tr>
<td>g2_{mw}</td>
<td>8.65</td>
<td>$81 \times 51 \times 2$</td>
</tr>
<tr>
<td>g3_{mw}</td>
<td>8.65</td>
<td>$161 \times 101 \times 2$</td>
</tr>
</tbody>
</table>
Table B.3: Main observations on uncertainty sources

<table>
<thead>
<tr>
<th>Uncertainty sources</th>
<th>Observations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discretization error</td>
<td>Both internal and external flow examples show that grid convergence is not achieved with grid levels that have moderate mesh sizes. When the flow-field includes shocks with substantial flow separation, highly refined grids, which are beyond the grid levels we use in this study, are needed for spatial convergence. Even with the finest mesh level we can afford, achieving asymptotic convergence is not certain. For the transonic diffuser case, the discretization error magnitudes are different for cases with different turbulence models, when nozzle efficiency results with the same limiter and grid level are compared at each shock condition. This indicates the effect of the turbulence model on grid convergence and implies that the magnitudes of numerical errors are influenced by the physical models used.</td>
</tr>
<tr>
<td>Error in geometry representation</td>
<td>For the transonic diffuser case, the main source of the discrepancy between the CFD results of the original geometry and the experiment upstream of the shock is the error in the geometry representation. Downstream of the shock, wall pressure results obtained with the same turbulence model and limiter are approximately the same regardless of the geometry used. This may imply that the difference between the experiment and the CFD results downstream of the shock is more likely due to the turbulence models when the finest grid levels are used to minimize the contribution of the discretization error.</td>
</tr>
<tr>
<td>Turbulence models</td>
<td>The strong and the weak shock results show that for each flow condition, the highest accuracy in terms of the wall pressure distributions is obtained with a different turbulence model, although the Sp-Al model gives reasonable results for both shock conditions. Uncertainties associated with the turbulence models interact strongly with the discretization errors. In some cases, numerical errors and the physical modelling uncertainties may cancel each other, and the closest results to the experiment can be obtained at intermediate grid levels.</td>
</tr>
<tr>
<td>Downstream boundary condition</td>
<td>Changing the location of the exit station or changing the exit pressure ratio affect the strength and the location of the shock. For the strong shock case, the size of the separation bubble is also affected by the same factors.</td>
</tr>
</tbody>
</table>
Table B.4: Discretization error results of the transonic diffuser case obtained with the original geometry. The cases presented in this table exhibit monotonic convergence with the refinement of the mesh size. For each case with a different turbulence model, limiter, and exit pressure ratio, the approximation to the exact value of $n_{eff}$ is denoted by $(\tilde{n}_{eff})_{\text{exact}}$ and the discretization error at a grid level $k$ is calculated by $\text{error}(\%) = \left| \frac{(n_{eff})_k - (\tilde{n}_{eff})_{\text{exact}}}{(n_{eff})_{\text{exact}}} \times 100 \right|.$

<table>
<thead>
<tr>
<th>turbulence model</th>
<th>limiter</th>
<th>$P_e/P_{0i}$</th>
<th>$\bar{p}$</th>
<th>$(\tilde{n}<em>{eff})</em>{\text{exact}}$</th>
<th>grid level</th>
<th>discretization error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sp-Al</td>
<td>Van Albada</td>
<td>0.72</td>
<td>1.528</td>
<td>0.71830</td>
<td>g1</td>
<td>9.820</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g2</td>
<td>4.505</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g3</td>
<td>1.562</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g4</td>
<td>0.542</td>
</tr>
<tr>
<td>Sp-Al</td>
<td>Min-Mod</td>
<td>0.72</td>
<td>1.322</td>
<td>0.71590</td>
<td>g1</td>
<td>14.298</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g2</td>
<td>6.790</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g3</td>
<td>2.716</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g4</td>
<td>1.086</td>
</tr>
<tr>
<td>Sp-Al</td>
<td>Van Albada</td>
<td>0.82</td>
<td>1.198</td>
<td>0.80958</td>
<td>g1</td>
<td>6.761</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g2</td>
<td>3.507</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g3</td>
<td>1.528</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g4</td>
<td>0.666</td>
</tr>
<tr>
<td>Sp-Al</td>
<td>Min-Mod</td>
<td>0.82</td>
<td>1.578</td>
<td>0.81086</td>
<td>g1</td>
<td>8.005</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g2</td>
<td>3.539</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g3</td>
<td>1.185</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g4</td>
<td>0.397</td>
</tr>
<tr>
<td>$k$-$\omega$</td>
<td>Van Albada</td>
<td>0.82</td>
<td>1.980</td>
<td>0.82962</td>
<td>g1</td>
<td>3.514</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g2</td>
<td>1.459</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g3</td>
<td>0.370</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g4</td>
<td>0.094</td>
</tr>
<tr>
<td>$k$-$\omega$</td>
<td>Min-Mod</td>
<td>0.82</td>
<td>1.656</td>
<td>0.82889</td>
<td>g1</td>
<td>4.432</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g2</td>
<td>1.452</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g3</td>
<td>0.461</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>g4</td>
<td>0.146</td>
</tr>
</tbody>
</table>
Table B.5: Top wall orthogonal distance error $\hat{E}_n$ calculated upstream of the experimental shock location (UESL) for each case obtained with the original geometry. Scaled error values $\hat{E}_n$ were obtained by $\hat{E}_n = \frac{E_n}{(E_n)_{max}} \times 100$ where $(E_n)_{max}$ is the maximum $E_n$ value calculated downstream of the experimental shock location (DESL) for the strong shock case with Grid 4, Min-Mod limiter, and the $k$-\(\omega\) turbulence model.

<table>
<thead>
<tr>
<th>$P_e/P_{bi}$</th>
<th>Grid</th>
<th>Sp-Al, Min-Mod</th>
<th>Sp-Al, Van Albada</th>
<th>$k$-(\omega), Min-Mod</th>
<th>$k$-(\omega), Van Albada</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.72</td>
<td>g1</td>
<td>25.6</td>
<td>26.5</td>
<td>27.3</td>
<td>28.2</td>
</tr>
<tr>
<td>0.72</td>
<td>g2</td>
<td>23.5</td>
<td>24.0</td>
<td>26.1</td>
<td>25.8</td>
</tr>
<tr>
<td>0.72</td>
<td>g3</td>
<td>23.9</td>
<td>24.0</td>
<td>26.3</td>
<td>26.2</td>
</tr>
<tr>
<td>0.72</td>
<td>g4</td>
<td>25.8</td>
<td>23.8</td>
<td>27.3</td>
<td>27.1</td>
</tr>
<tr>
<td>0.82</td>
<td>g1</td>
<td>27.3</td>
<td>29.3</td>
<td>28.9</td>
<td>31.1</td>
</tr>
<tr>
<td>0.82</td>
<td>g2</td>
<td>27.1</td>
<td>27.5</td>
<td>28.0</td>
<td>28.4</td>
</tr>
<tr>
<td>0.82</td>
<td>g3</td>
<td>27.7</td>
<td>27.8</td>
<td>28.4</td>
<td>28.5</td>
</tr>
<tr>
<td>0.82</td>
<td>g4</td>
<td>27.6</td>
<td>27.6</td>
<td>28.2</td>
<td>28.2</td>
</tr>
</tbody>
</table>
Table B.6: Top wall orthogonal distance error $\hat{E}_n$ calculated downstream of the experimental shock location (DESL) for each case obtained with the original geometry.

<table>
<thead>
<tr>
<th>$P_e/P_{bi}$</th>
<th>Grid</th>
<th>Sp-Al, Min-Mod</th>
<th>Sp-Al, Van Albada</th>
<th>$k-\omega$, Min-Mod</th>
<th>$k-\omega$, Van Albada</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.72</td>
<td>g1</td>
<td>81.2</td>
<td>64.4</td>
<td>85.6</td>
<td>74.6</td>
</tr>
<tr>
<td>0.72</td>
<td>g2</td>
<td>52.3</td>
<td>48.9</td>
<td>89.9</td>
<td>83.7</td>
</tr>
<tr>
<td>0.72</td>
<td>g3</td>
<td>35.0</td>
<td>34.5</td>
<td>90.1</td>
<td>89.2</td>
</tr>
<tr>
<td>0.72</td>
<td>g4</td>
<td>27.8</td>
<td>27.9</td>
<td>100.0</td>
<td>97.8</td>
</tr>
<tr>
<td>0.82</td>
<td>g1</td>
<td>27.1</td>
<td>21.4</td>
<td>14.6</td>
<td>14.6</td>
</tr>
<tr>
<td>0.82</td>
<td>g2</td>
<td>11.3</td>
<td>10.9</td>
<td>14.6</td>
<td>14.3</td>
</tr>
<tr>
<td>0.82</td>
<td>g3</td>
<td>17.7</td>
<td>16.9</td>
<td>12.9</td>
<td>13.3</td>
</tr>
<tr>
<td>0.82</td>
<td>g4</td>
<td>21.2</td>
<td>20.8</td>
<td>10.8</td>
<td>10.7</td>
</tr>
</tbody>
</table>
Table B.7: Nozzle efficiency values obtained with different grid levels, limiters, turbulence models, geometries, and boundary conditions.

<table>
<thead>
<tr>
<th>turbulence model</th>
<th>limiter</th>
<th>grid level</th>
<th>strong shock</th>
<th>weak shock</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>original geometry</td>
<td>modified-wall geometry</td>
<td>extended geometry</td>
</tr>
<tr>
<td>k-\omega</td>
<td>Min-mod</td>
<td>1</td>
<td>0.81113</td>
<td>0.80556</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2</td>
<td>0.79362</td>
<td>0.79640</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td>0.78543</td>
<td>0.78886</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4</td>
<td>0.79007</td>
<td></td>
</tr>
<tr>
<td>k-\omega</td>
<td>Van Albada</td>
<td>1</td>
<td>0.78820</td>
<td>0.78333</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2</td>
<td>0.78199</td>
<td>0.78439</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td>0.78310</td>
<td>0.78661</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4</td>
<td>0.78788</td>
<td></td>
</tr>
<tr>
<td>Sp-Al</td>
<td>Min-mod</td>
<td>1</td>
<td>0.81827</td>
<td>0.81562</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2</td>
<td>0.76452</td>
<td>0.76479</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td>0.73535</td>
<td>0.73402</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4</td>
<td>0.72369</td>
<td></td>
</tr>
<tr>
<td>Sp-Al</td>
<td>Van Albada</td>
<td>1</td>
<td>0.78885</td>
<td>0.78647</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2</td>
<td>0.75067</td>
<td>0.74850</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td>0.72933</td>
<td>0.72569</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4</td>
<td>0.72220</td>
<td>0.73268</td>
</tr>
</tbody>
</table>

Note: The table shows the nozzle efficiency values obtained with different grid levels, limiters, turbulence models, geometries, and boundary conditions. The values are given in the form of ratios of pressure, $P_e/P_{0i}$, and extended geometry parameters $P_e/P_{0i}$. The table includes columns for strong shock and weak shock conditions, with values rounded to two decimal places.
### Table B.8: Main observations on the uncertainty in nozzle efficiencies

<table>
<thead>
<tr>
<th>Shock type</th>
<th>Observations on uncertainties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strong shock case</td>
<td>The range of variation in nozzle efficiency results is much larger than that observed in the weak shock case. The maximum variation is about 10% for the strong shock case, and 4% for the weak shock case, when the results of the original geometry are compared.</td>
</tr>
<tr>
<td></td>
<td>Magnitude of the discretization errors is larger than that of the weak shock case. The discretization errors at grid level 2 can be up to 6% for the strong shock case.</td>
</tr>
<tr>
<td></td>
<td>Relative uncertainty due to the selection of the turbulence model can be larger than that due to discretization errors depending upon the grid level used. This uncertainty can be as large as 9% at grid level 4.</td>
</tr>
<tr>
<td></td>
<td>The contribution of the error in geometry representation to the overall uncertainty is negligible compared to the other sources of uncertainty.</td>
</tr>
<tr>
<td>Weak shock case</td>
<td>Discretization error is the dominant source of uncertainty. The maximum value of the discretization error is 3.5%, whereas the maximum value of turbulence model uncertainty is about 2%.</td>
</tr>
<tr>
<td></td>
<td>The nozzle efficiency values are more sensitive to the exit boundary conditions and associated error magnitudes can be larger than from other sources. The difference between the results from original geometry and the extended geometry can be as large as 7% when the exit pressure ratio of 0.8368 is used.</td>
</tr>
<tr>
<td></td>
<td>The contribution of the error in geometry representation to the overall uncertainty can be up to 1.5%.</td>
</tr>
</tbody>
</table>
Table B.9: Discretization errors calculated by using the results of different grid levels for the transonic diffuser case with the original geometry, Sp-Al turbulence model, and the Min-Mod limiter.

<table>
<thead>
<tr>
<th>grid levels used</th>
<th>( \tilde{p} )</th>
<th>((\tilde{n}<em>{eff})</em>{exact})</th>
<th>grid level</th>
<th>error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>for ( \tilde{p} ): g2, g3, and g4</td>
<td>1.322</td>
<td>0.71590</td>
<td>g1</td>
<td>14.298</td>
</tr>
<tr>
<td>for ((\tilde{n}<em>{eff})</em>{exact}) : g3 and g4</td>
<td>1.849</td>
<td>0.71921</td>
<td>g1</td>
<td>13.774</td>
</tr>
<tr>
<td>for ( \tilde{p} ): g3, g4, and g5</td>
<td>1.322</td>
<td>0.71590</td>
<td>g2</td>
<td>6.790</td>
</tr>
<tr>
<td>for ((\tilde{n}<em>{eff})</em>{exact}) : g3 and g4</td>
<td>1.849</td>
<td>0.71921</td>
<td>g2</td>
<td>6.300</td>
</tr>
<tr>
<td>g4</td>
<td>2.716</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>g5</td>
<td>0.634</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>g4</td>
<td>1.086</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>g5</td>
<td>0.623</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>g5</td>
<td>0.173</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
B.7 Figures of Appendix B

Figure B.1: Original geometry, Grid 2 (top), and extended geometry, Grid $2_{ext}$ (bottom), used in the transonic diffuser computations. The flow is from left to right, in the positive $x$-direction. The $y$-direction is normal to the bottom wall. All dimensions are scaled by the throat height, $h_t$. The throat section, which is the minimum cross-sectional area of the channel, is located at $x/h_t = 0.0$. Both geometries have the inlet stations located at $x/h_t = -4.04$. 
Figure B.2: Velocity contours, streamlines, and the top wall pressure distributions of the weak and the strong shock cases.
Figure B.3: Normalized $L^2$ residual of the energy equation for the case with the Sp-Al turbulence model, Van Albada, and Min-Mod limiters at $P_e/P_{0i} = 0.72$ obtained with the original geometry. Normalization is done with the initial value of the residual.
Figure B.4: Convergence history of the nozzle efficiency at different grid levels for the Sp-Al, Min-Mod, strong shock case obtained with the original geometry. (The nozzle efficiency values are monitored at every 50 cycles starting from iteration number 10000 for Grid 5)
Figure B.5: Nozzle efficiencies obtained with different grid levels, turbulence models, limiters, geometries, and boundary conditions for the strong shock case (A) and the weak shock case (B).
Figure B.6: Nozzle efficiency vs. exit pressure ratio for different grids obtained with the original geometry, Sp-Al and $k$-$\omega$ turbulence models, and the Min-Mod limiter.
Figure B.7: Mach number values at the upstream of the shock ($x/h_{t} = -1.5$), and downstream of the shock ($x/h_{t} = 8.65$, the exit plane) for different grids obtained with the original geometry, Sp-Al and $k-\omega$ turbulence models, Min-Mod and Van Albada limiters. The values of $y/h_{t}$ correspond to the midpoints of the local channel heights.
Figure B.8: Error distribution in $y/h_t$ for the upper wall of the modified-wall diffuser geometry at the data points measured in the experiments. The maximum error is approximately 7% and observed upstream of the throat, at $x/h_t = -1.95$. Starting from $x/h_t = 1.2$, the error is approximately constant with an average value of 0.9%.
Figure B.9: Upper wall contours of the original and the modified-wall diffuser geometry in the vicinity of the throat location.
Figure B.10: Top wall pressure distributions obtained with the original and the modified-wall geometry for the strong shock case (the results of the Sp-Al model, Min-Mod limiter, and Grids g2 and $g_{mw}^2$ are shown).
Figure B.11: Top wall pressure distributions obtained with the original and the modified-wall geometry for the weak shock case (the results of the Sp-Al model, Min-Mod limiter, and Grids g2 and g2_{mw} are shown).
Figure B.12: Streamline patterns of the separated flow region obtained with different versions of the diffuser geometry and exit pressure ratios for the strong shock case.
Figure B.13: Comparison of the separation bubbles obtained with different versions of the diffuser geometry and exit pressure ratios for the strong shock case.
Figure B.14: Top wall pressure distributions obtained with different versions of the diffuser geometry and exit pressure ratios for the strong shock case (the results of the Sp-Al model, Van Albada limiter, and Grids g3 and g3_{ext} are shown).
Vita

Serhat Hosder was born on November 4, 1975 in Ankara, the capital of Turkey. In 1993, he graduated from Bornova Anatolian High School (BAL), Izmir. In the same year, he began his undergraduate study in Aeronautics and Astronautics Department of Istanbul Technical University (ITU), Istanbul. In 1997, he graduated from the same school with a Bachelor degree in Aeronautical Engineering. He received his M.S. degree in Aerospace Engineering from Virginia Tech in June 2001. He earned his PhD degree in Aerospace Engineering from the same university in September 2004. After getting his PhD degree, he started to work for Techsburg Inc. in Blacksburg, as a Principal Research Engineer.