A thesis submitted in accordance with the regulations for the
degree of Doctor of Philosophy

By
Nurul Muiz Murad

School of Engineering and Industrial Science
Swinburne University of Technology, Melbourne, Australia

February, 2009
ABSTRACT

Vortex generation behind the A-pillar region due to airflow separation leads to aero-acoustics generation. The magnitude and intensity of the vortex and hence aero-acoustics activities are further enhanced when vehicle are exposed to crosswind especially when travelling on a highway. The objective of this project is to develop a computational fluid dynamic (CFD) and computational aero-acoustics (CAA) model to best simulate aerodynamic flow and aero-acoustics propagation behind the A-pillar region of simplified vehicle with varying windshield radii under various yaw conditions. The CFD model will then be used to investigate and better understand the aerodynamic and aero-acoustics distribution behaviour surrounding area of the vehicle A-pillar region. The simplified vehicle model used was of 40% scale. Models investigated consist of three models of different circular windshield/A-pillar radii and two models of with sharp A-pillar edges with different windshield slant angle. Models used in this project were subjected to 0°, 5°, 10° and 15° yaw angles. The models were modelled under laboratory conditions, exposed to boundary inlet velocities of 60, 100 and 140 km/h.

The development of the CFD model consists of first investigating and selecting the best grids configurations for both the circular and sharp edge A-pillar models at various respective yaw angles. The process was then to select the best turbulence and near wall model for the CFD model. The grids, turbulence and near wall models selected for comparison were investigated from commercial CFD software’s FLUENT and SWIFT AVL. The final stage in the development of the numerical model was to develop a CAA model for the aero-acoustics modelling. The final grid configuration selected for the CFD models in this project was the polyhedral grids from SWIFT. The final selection of turbulence and near wall model selected was the standard $k – \varepsilon$ turbulence model and the near wall model of Chieng and Launder (1980) for the circular models at 0° yaw. For all the other models at various yaw angles, the turbulence and near wall model of choice was the RSM with the WEB near wall model. Validation of the final CFD model against the experimental data of Alam (2000) resulted in good correlations.
The CAA model developed for this project was conducted using SWIFT CAA and was conducted only for the circular models due to their better correlations with experimental data. From the CFD and CAA numerical models developed, investigation was then conducted in determining the source and mechanism of vortex generation behind the A-pillar and the overall physical shape and turbulence flow characteristics of the A-pillar vortex itself. The CAA investigation focused in determining the transient behaviour of the models and also the acoustical behaviour on the A-pillar surface and surrounding region.

The results obtained from the CFD analysis shows that the source of vortex separation behind the A-pillar region originated from the junction of the A-pillar base, the A-pillar apex and the front side window and roof junction. The mechanism of flow separation was due to trailing edge separation. The shape of the vortices that takes place took a physical form of either a two-dimensional quasi-elongated oval, a mixture of two and three-dimensional mixture of a quasi circular and cone shaped helical vortex or a three-dimensional vertically elongated cone shape helical vortex propagating downstream to the flow. The various geometrical configurations of the windshield radii and slant angle determined the vortex size, magnitude and intensity behind the A-pillar region when exposed to yawed or un-yawed position.

Hybrid SWIFT CAA results showed good correlation when compared to results obtained by Alam (2000). The transient progression for each investigated scale model shows that the circular and sharp edge models investigated will reach a faster steady state condition with increasing windshield radii and when exposed to un-yawed condition due to a reduced turbulent activities behind the A-pillar region. Results show that the OASPL magnitude is higher on vehicle subjected to yawing conditions. Results also show that OASPL magnitude on the vehicle surface decreases with increases windshield radii. The aerodynamic noise generation decreases as it moves away from the vehicle surface. Overall, mean surface OASPL magnitude at the vortex source region (A-pillar Base Junction, A-pillar apex and Roof) is slightly higher compared to the overall mean OASPL magnitude on the surface of the vehicle.
ACKNOWLEDGEMENTS

Bismillah-ir Rahman-ir Rahim, Alhamdulillah syukur ke hadrat Allah s.w.t. kerana dengan limpah dan kurnia-Nya akhirnya dapat juga saya menghabiskan thesis ini.

Saya ingin mengambil kesempatan ini untuk mengucapkan ribuan terima kasih yang tidak terhingga buat kedua ibu bapa saya, Haji Murad bin Haji Ahmad dan Hajjah Alisma binti Haji Sarudin. Berdirinya saya pada hari ini sebagai seorang manusia yang sempurna adalah berkat doa serta hasil usaha mereka membimbing dan memberi tunjuk ajar kepada saya.

Al-Fatihah buat Allahyarhammah Hajja h Daimah binti Haji Daud, moyang saya yang bersusah payah menjaga saya semasa kecil. Semoga rohnya sentiasa dicucuri rahmat. Amin.

Terima kasih yang tidak terhingga buat nenek saya, Hajjah Sabedar binti Haji Abdul Munaf dan makandak saya, Hajjah Siti Harani binti Haji Sarudin. Saya tidak akan dapat membalas jasa mereka menjaga dan mendidik dari kecil sehingga dewasa. Terima kasih juga buat pakcik serta makkik saya, Pak Long dan Mak Long, Pak Gadang dan Mak Gadang, Pak Ngah dan Mak Ngah, Pak Lang dan Mak Lang, Pak Uteh dan Mak Uteh, Pak Anjang dan Mak Anjang serta Pak Kecik dan Mak Kecik. Kasih sayang serta tunjuk ajar mereka sedikit sebanyak mencorak perwatakan saya pada hari ini.

Terima kasih buat adik-adik saya, Nur Meliza binti Haji Murad, Nurul Muhriz bin Haji Murad dan Nurul Mahfuz bin Haji Murad. Hadir mereka dalam hidup saya telah memberi saya perangsang untuk terus maju dan menjadi yang terbaik buat contoh untuk mereka ikuti. Tidak lupa juga, saya tujuan kejayaan saya buat sepupu-sepupu saya serta rakan-rakan saya di seluruh Malaysia dan Australia terutamanya rakan-rakan lama di MTD, AUSMAT dan Dewan Malaysia.

Terima kasih buat supervisor saya Dr. Jamal Naser dan Dr. Simon Watkins. Terutama buat Dr. Jamal Naser yang telah memberi kepercayaan, motivasi serta
bimbingan buat saya untuk menghabiskan thesis ini. Dr. Jamal Naser bukan sahaja menjadi supervisor malah mentor dan kawan baik saya.

Terima kasih buat rakan seperjuangan saya, Dr. Firoz Alam, Dr. James Hart dan Dr. Gregory Chawynski yang telah menolong saya dan membimbing saya sepanjang perjuangan saya menyiapkan thesis ini.

Tidak lupa, setingga-tinggi penghargaan dan ucap terima kasih buat mentor dan rakan baik saya di Melbourne, Encik Ahmad Fuad Mansor yang telah berada disisi saya disaat susah dan senang. Pertolongan, nasihat serta bimbingan beliau akan saya kenang sampai bila-bila.

Terima kasih buat kerajaan Malaysia yang telah menaruh kepercayaan kepada saya dengan menghantar saya ke bumi Australia untuk melanjutkan pelajaran dalam peringkat sarjana muda. Tanpa inisiatif dari mereka, semua ini tidak akan menjadi kenyataan.

Akhir sekali, terima kasih saya ucapkan buat saudari Jasmin binti Mohd Ramli yang sentiasa menjadi sumber inspirasi saya dalam mengharungi ranjau hidup yang penuh mencabar.

Nurul Muiz Murad
Ogos 2008
DECLARATION OF ORIGINALITY

I, Nurul Muiz Murad, hereby declare that this thesis contains no material, which has been accepted for the award of any other degree or diploma in any university or institute of education. To the best of my knowledge and belief, no material in this thesis has been previously published or written by another person except where due references is made in the body of the thesis.

Signed  ………………………………….

Nurul Muiz Murad

August, 2008
TABLE OF CONTENTS

Abstract ii
Acknowledgements iv
Declaration of Originality vi
Table of Contents vii
List of Figures xi
List of Tables xxiii
Nomenclature xxv
List of Abbreviations and Acronyms xxvii

Chapter One: Introduction & Literature Review 1
1.1 History of Vehicle Aerodynamics: A General Background 1
1.2 Airflow around a Ground Vehicle 2
1.3 Overview on Sound and Noise 6
1.4 Problems associated with Vehicle Vortex Flow 10
1.5 Vehicle Noise 13
1.6 Mechanism of Aerodynamic Generation 16
1.7 Ways of Reducing A-Pillar Aerodynamic Noise 22
1.8 Vehicle Aerodynamics and Aeroacoustics: Numerical and Computational Evaluation Methods 24
1.9 Literature Review on A-Pillar Aerodynamics and Aeroacoustics 27
1.10 Conclusions and Evaluation from Previous Research Work 38
1.11 Research Project Motivation, Scope and Proposed Methodology 42
1.12 Objectives of PhD Project 43
1.13 Thesis Layout 44

Chapter Two: Governing Equations and Boundary Conditions 46
2.1 Turbulence and Early Works of Turbulence Modelling 46
2.2 Governing Transport Equation and Turbulence Models 48
2.3 Algebraic (zero equation) Turbulence Models 51
2.4 One Equation Turbulence Models 52
2.5 Two Equation Turbulence Models 54
2.6 Deficiencies of the Two Equation Turbulence Model 63
2.6.1 Pressure Gradient Effects 63
2.6.2 Effect of Rapid change of Mean Strain Rate and Streamline Curvature 64
2.7 Reynolds Stress Turbulence Models 65
2.8 Direct Numerical and Large Eddy Simulation 73
2.9 CFD Near Wall Treatment and Boundary Conditions  
2.10 Wall Function Approach  
2.11 Low Reynolds Number Model approach  
2.12 Boundary Conditions  
2.13 Computational Aeroacoustics  
2.14 Lighthill Acoustic Analogy Method  
2.15 Kirchoff Method  
2.16 Perturbation Method  
2.17 Linearized Euler Equation Method  

Chapter Three: Methodology  
3.1 General CFD Approach Process  
3.2 Accuracy Factors and Errors Associated with CFD  
3.3 CFD Grid Generation and Discretization Methods  
3.4 CFD Numerical Schemes  
3.5 Segregated and Coupled Solver  
3.6 Near Wall Models and Turbulence Models  
3.7 CAD Model Geometry and Boundary Conditions Input  

Chapter Four: RANS CFD of A-Pillar Aerodynamics  
4.1 Objective and Scope of this Chapter  
4.2 CFD Model Development  
  4.2.1 Grid Feasibility Study – Generation Technique  
  4.2.2 Grid Feasibility Study – Grid Refinement & Independency Procedure  
  4.2.3 Grid Feasibility Study – Validation with Experimental Results at 0° Yaw  
  4.2.4 Near Wall Model and Turbulence Model Feasibility Study  
   4.2.4.1 Circular Models at 0° Yaw - Near Wall Model and Turbulence Model Feasibility Study  
   4.2.4.2 Circular Models at 5°, 10° and 15° Yaw – Near Wall Model and Turbulence Model Feasibility Study  
   4.2.4.3 Sharp Edge Models at 0°, 5°, 10° and 15° Yaw – Near Wall Model and Turbulence Model Feasibility Study  
4.3 Circular Models at 0° Yaw – Results and Discussion  
4.4 Circular Models at 5° yaw – Results and Discussion  
4.5 Circular Models at 10° and 15° Yaw – Results and Discussion  
4.6 RE Model at 0° Yaw – Results and Discussion  
4.7 SL Model at 0° Yaw – Results and Discussion  
4.8 RE Model at 5°, 10° and 15° Yaw – Results and Discussion  
4.9 Slanted Model at 5°, 10° and 15° Yaw – Results and Discussion  
4.10 General Discussion
Chapter Five: Computational Aeroacoustics (CAA) Simulations

5.1 Introduction to the Hybrid SWIFT CAA Approach

5.2 Methodology of the Hybrid SWIFT CAA Approach

5.3 Objectives & Scope of using Hybrid SWIFT CAA Approach: Application to this Research Project

5.3.1 Objectives of Chapter 5

5.3.2 Scope of Chapter 5

5.4 Hybrid SWIFT CAA Results

5.4.1 Hybrid SWIFT CAA & Experimental Validation - SE Model, 0° & 15° Yaw

5.4.2 Hybrid SWIFT CAA & Experimental Validation - Semi Model, 0° & 15° Yaw

5.4.3 Hybrid SWIFT CAA & Experimental Validation – LE Model, 0° & 15° Yaw

5.4.4 Hybrid SWIFT CAA & Experimental Validation – RE Model, 0° & 15° Yaw

5.4.5 Hybrid SWIFT CAA & Experimental Validation – SL Model, 0° & 15° Yaw

5.4.6 Aeroacoustics Behaviour during Transient Condition – SE Model, 0° & 15° Yaw

5.4.7 Aeroacoustics Behaviour during Transient Condition – Semi Model, 0° & 15° Yaw

5.4.8 Aeroacoustics Behaviour during Transient Condition – LE Model, 0° & 15° Yaw

5.4.9 Aeroacoustics Behaviour during Transient Condition – RE Model, 0° & 15° Yaw

5.4.10 Aeroacoustics Behaviour during Transient Condition – SL Model, 0° & 15° Yaw

5.4.11 CAA Behaviour & Distribution behind A-pillar Region – SE Model, 0° Yaw

5.4.12 CAA Behaviour & Distribution Behind A-pillar Region – SE Model, 15° Yaw

5.4.13 CAA Behaviour & Distribution Behind A-pillar Region – Semi Model, 0° Yaw

5.4.14 CAA Behaviour & Distribution Behind A-pillar Region – Semi Model, 15° Yaw

5.4.15 CAA Behaviour & Distribution Behind A-pillar Region – LE Model, 0° Yaw

5.4.16 CAA Behaviour & Distribution Behind A-pillar Region – LE Model, 15° Yaw

5.5 Discussion of Hybrid SWIFT CAA Results

5.5.1 Comparison between Hybrid SWIFT CAA & Experimental Results – Cp RMS Pressure

5.5.2 Comparison between Hybrid SWIFT CAA & Experimental Results – PSD Distribution

5.5.3 Aero-acoustics Behaviour during Transient Conditions

5.5.4 CAA Behaviour Behind the A-pillar Region
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chapter Six: Conclusions &amp; Recommendations</td>
<td>306</td>
</tr>
<tr>
<td>6.1 Conclusions from Chapter 4</td>
<td>306</td>
</tr>
<tr>
<td>6.2 Conclusions from Chapter 5</td>
<td>310</td>
</tr>
<tr>
<td>6.3 Further Recommendations</td>
<td>313</td>
</tr>
<tr>
<td>References</td>
<td>315</td>
</tr>
<tr>
<td>Bibliography</td>
<td>330</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

Figure 1.1. Areas of Separation around a Vehicle 4
Figure 1.2. Slanted A-pillar Vortex flow 5
Figure 1.3. High Pressure Zone in a Vortex Flow behind a Backward-Facing Step 7
Figure 1.4. Fletcher Munsen Curve 8
Figure 1.5. Forced and Free Vortex Velocity Distribution 11
Figure 1.6. Two method of Leak Noise Transmission 18
Figure 1.7. Mechanism of Cavity Noise Transmission 20
Figure 1.8. Wind Rush Noise Transmission around Vehicle 22
Figure 3.1: Simplified Vehicle Model Geometry with Varying A-pillar Windshield Radius 116
Figure 3.2: Models in 0°, 5°, 10° and 15° yaw position 117
Figure 3.3: Small Ellipsoidal Model in a yaw position within Computational Domain in AVL 117
Figure 3.4: Slanted Edge Model within Computational Domain in FLUENT 118
Figure 3.5: Semi Circular Model within Computational Domain in AVL 119
Figure 3.6: Slanted Edge Model with Multi-block volumes in GAMBIT 119
Figure 4.1: Slanted Edge Model with Multi-block volumes in GAMBIT 125
Figure 4.2: Fully Structured Hexahedral Grids Layout in GAMBIT 126
Figure 4.3: Grid Skewness Quality of the Fully Structured Hexahedral Grids in GAMBIT 126
Figure 4.4: Hybrid Grids Layout in GAMBIT 127
Figure 4.5: Grid Skewness Quality of the Hybrid Grids in GAMBIT 128
Figure 4.6: Unstructured Tetrahedral Grids Layout in GAMBIT 128
Figure 4.7: Grid Skewness Quality of the Unstructured Tetrahedral Grids in GAMBIT 129
Figure 4.8: ACT Polyhedral Grids Layout in AVL generated using Fame Hybrid 130
Figure 4.9: Before and After Solution Adaptive Refinement for Semi Circular Model meshed with unstructured Tetrahedral 133
Grids in GAMBIT

Figure 4.10: Boundary Layer Grids for the Slanted Edge Model generated using the Hybrid Grids Method in GAMBIT

Figure 4.11: Grid Independency Test for the Semi Circular Model in AVL at 0° Yaw

Figure 4.12: Grid Independency Test for the Semi Circular Model in AVL at 15° Yaw

Figure 4.13: Grid Independency Test Results for the Semi Circular Model in FLUENT at 0° Yaw

Figure 4.14: Grid Independency Test Results for the Semi Circular Model in AVL at 0° Yaw

Figure 4.15: Comparison between Hexahedral, Hybrid and Tetrahedral Grid Generation Method in FLUENT at 0° Yaw

Figure 4.16: Grid Independency Test Results for the Slanted Edge Model in AVL at 0° Yaw with Standard Wall Function

Figure 4.17: Comparison between Hybrid and Polyhedral Grids at High and Low Reynolds Number Conditions

Figure 4.18: SE Model, 0° Yaw

Figure 4.19: Semi Model, 0° Yaw

Figure 4.20: LE Model, 0° Yaw

Figure 4.21: SE Model, 5° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.22: SE Model, 10° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.23: SE Model, 15° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.24: Semi Model, 5° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.25: Semi Model, 10° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.26: Semi Model, 15° Yaw, Bottom Row, Turbulence Model Comparison
Figure 4.27: LE Model, 5° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.28: LE Model, 10° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.29: LE Model, 15° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.30: SE Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.31: SE Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.32: SE Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.33: Semi Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.34: Semi Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.35: Semi Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.36: LE Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.37: LE Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.38: LE Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.39: SL Model, 0° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.40: SL Model, 0° Yaw, Top Row, Turbulence Model Comparison

Figure 4.41: SL Model, 0° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.42: SL Model, 0° Yaw, Top Row, Turbulence Model Comparison

xiii
Figure 4.43: Slanted Edge Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.44: Slanted Edge Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.45: Slanted Edge Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.46: Rectangular Model, 0° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.47: Rectangular Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.48: Rectangular Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.49: Rectangular Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.50: SE Model, 0° Yaw, External Surface Streamline, Front View
Figure 4.51: SE Model, 0° Yaw, External Surface Streamline, Top View
Figure 4.52: SE Model, 0° Yaw, Top View, Turbulent Velocity
Figure 4.53: SE Model, 0° Yaw, Front View, Velocity Vector
Figure 4.54: SE Model, 0° Yaw, Surface Streamline
Figure 4.55: SE Model at 0° Yaw, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)
Figure 4.56: Semi Model, 0° Yaw, Top View, Turbulent Velocity
Figure 4.57: LE Model, 0° Yaw, Top View, Turbulent Velocity
Figure 4.58: SE Model, 5° Yaw, Leeward Region, External Flow Streamline
Figure 4.59: SE Model, 5° Yaw, Windward Region, External Flow Streamline
Figure 4.60: Semi Model, 5° Yaw, Front View, External Flow Streamline
Figure 4.61: LE Model, 5° Yaw, Front View, External Flow Streamline
Figure 4.62: SE Model, 5° Yaw, Front View, Velocity Vector
Figure 4.63: SE Model, 5° Yaw, Top View, Turbulent Velocity
Figure 4.64: SE Model, 5° Yaw, Reynolds Stresses Component, Leeward and Windward Region of the A-pillar
Figure 4.65: SE Model, 10° Yaw, Front View, External Surface Streamline
Figure 4.66: SE Model, 15° Yaw, Leeward View, External Surface Streamline
Figure 4.67: Semi Model, 10° Yaw, Front View, External Surface Streamline
Figure 4.68: LE Model, 10° Yaw, Front View, External Surface Streamline
Figure 4.69: Semi Model, 15° Yaw, Front View, External Surface Streamline
Figure 4.70: LE Model, 15° Yaw, Front View, External Surface Streamline
Figure 4.71: SE Model, 15° Yaw, Leeward Region, Surface Flow Streamline
Figure 4.72: SE Model at 15° Yaw in the Leeward Region, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)
Figure 4.73: SE Model, 15° Yaw, Windward Region, Surface Flow Streamline
Figure 4.74: SE Model at 15° Yaw in the Windward Region, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)
Figure 4.75: SE Model, 10° Yaw, Top View, Turbulent Velocity
Figure 4.76: SE Model, 10° Yaw, Front View, Velocity Vector
Figure 4.77: SE Model, 15° Yaw, Top View, Turbulent Velocity
Figure 4.78: SE Model, 15° Yaw, Front View, Velocity Vector
Figure 4.79: SE Model, 10° Yaw, Reynolds Stresses Component, Leeward and Windward Region of the A-pillar
Figure 4.80: SE Model, 15° Yaw, Reynolds Stresses Component, Leeward and Windward Region of the A-pillar
Figure 4.81: Rectangular Model, 0° Yaw, Front View, External Flow Streamline
Figure 4.98: Rectangular Model, 15° Yaw, Frontal External Streamline Airflow

Figure 4.99: Rectangular Model, 15° Yaw, Turbulent Velocity

Figure 4.100: Rectangular Model, 15° Yaw, Leeward Surface Streamline Airflow

Figure 4.101: RE Model at 15° Yaw in the Leeward Region, Surface Streamline Visualisation using Wool Tuffs

Figure 4.102: Rectangular Model, 15° Yaw, Windward Surface Streamline Airflow

Figure 4.103: RE Model at 15° Yaw in the Windward Region, Surface Streamline Visualisation using Wool Tuffs

Figure 4.104: Slanted Model, 5° Yaw, Frontal External Streamline Airflow

Figure 4.105: Slanted Model, 10° Yaw, Frontal External Streamline Airflow

Figure 4.106: Slanted Model, 15° Yaw, Frontal External Streamline Airflow

Figure 4.107: Slanted Model, 15° Yaw, Top View, Turbulent Velocity

Figure 4.108: Slanted Model, 15° Yaw, Leeward Surface Streamline Airflow

Figure 4.109: SL Model at 15° Yaw in the Leeward Region, Surface Streamline Visualisation using Wool Tuffs

Figure 4.110: Slanted Model, 15° Yaw, Windward Surface Streamline Airflow

Figure 4.111: SL Model at 15° Yaw in the Windward Region, Surface Streamline Visualisation using Wool Tuffs

Figure 5.1 Unstructured Tetrahedral CAA Domain of Various Simplified Vehicle Model Created from AVL SWIFT and GAMBIT

Figure 5.2: Comparison of Cp RMS between Numerical and
Experimental Results, Small Ellipsoidal Model, 0° Yaw

Figure 5.3: Comparison of Cp RMS between Numerical and Experimental Results, Small Ellipsoidal Model, -15° Yaw

Figure 5.4: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Small Ellipsoidal Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8-kHz Frequency Region

Figure 5.5: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Small Ellipsoidal Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 6-kHz Frequency Region

Figure 5.6: Experimental Results of Spectral Energy Density Distribution for Small Ellipsoidal Model at -15°, 0° and +15° Yaw (After Alam, 2000)

Figure 5.7: Comparison of Cp RMS between Numerical and Experimental Results, Semi Circular Model, 0° Yaw

Figure 5.8: Comparison of Cp RMS between Numerical and Experimental Results, Semi Circular Model, -15° Yaw

Figure 5.9: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Semi Circular Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 1.4-kHz Frequency Region

Figure 5.10: Experimental Results of Spectral Energy Density Distribution for Semi Circular Model at -15°, 0° and +15° Yaw (After Alam, 2000)

Figure 5.11: Comparison of Cp RMS between Numerical and Experimental Results, Large Ellipsoidal Model, 0° Yaw

Figure 5.12: Comparison of Cp RMS between Numerical and Experimental Results, Large Ellipsoidal Model, -15° Yaw

Figure 5.13: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Large Ellipsoidal Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 1.2-kHz Frequency Region

Figure 5.14: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Large Ellipsoidal Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8.0-kHz Frequency Region
Figure 5.15: Experimental Results of Spectral Energy Density Distribution for Large Ellipsoidal Model at -15°, 0° and +15° Yaw (After Alam, 2000)

Figure 5.16: Comparison of Cp RMS between Numerical and Experimental Results, Rectangular Edge Model, 0° Yaw

Figure 5.17: Comparison of Cp RMS between Numerical and Experimental Results, Rectangular Edge Model, -15° Yaw

Figure 5.18: Comparison of Cp RMS between Numerical and Experimental Results, Rectangular Edge Model, +15° Yaw

Figure 5.19: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Rectangular Edge Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8.0-kHz Frequency Region

Figure 5.20: Experimental Results of Spectral Energy Density Distribution for Rectangular Edge Model at -15°, 0° and +15° Yaw (After Alam, 2000)

Figure 5.21: Comparison of Cp RMS between Numerical and Experimental Results, Slanted Edge Model, 0° Yaw

Figure 5.22: Comparison of Cp RMS between Numerical and Experimental Results, Slanted Edge Model, -15° Yaw

Figure 5.23: Comparison of Cp RMS between Numerical and Experimental Results, Slanted Edge Model, +15° Yaw

Figure 5.24: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Slanted Edge Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8.0-kHz Frequency Region

Figure 5.25: Experimental Results of Spectral Energy Density Distribution for Slanted Edge Model at -15°, 0° and +15° Yaw (After Alam, 2000)

Figure 5.26: Cp RMS Temporal Progression for Small Ellipsoidal Model, 0° Yaw, 60, 100 and 140 km/h

Figure 5.27: Cp RMS Temporal Progression for Small Ellipsoidal Model, 15° Yaw, 60, 100 and 140 km/h

Figure 5.28: Cp RMS Temporal Progression for Semi Circular xix
Figure 5.29: Cp RMS Temporal Progression for Semi Circular Model, 0° Yaw, 60, 100 and 140 km/h
Figure 5.30: Cp RMS Temporal Progression for Large Ellipsoidal Model, 0° Yaw, 60, 100 and 140 km/h
Figure 5.31: Cp RMS Temporal Progression for Large Ellipsoidal Model, 15° Yaw, 60, 100 and 140 km/h
Figure 5.32: Cp RMS Temporal Progression for Rectangular Edge Model, 0° Yaw, 60, 100 and 140 km/h
Figure 5.33: Cp RMS Temporal Progression for Rectangular Edge Model, 15° Yaw, 60, 100 and 140 km/h
Figure 5.34: Cp RMS Temporal Progression for Slanted Edge Model, 0° Yaw, 60, 100 and 140 km/h
Figure 5.35: Cp RMS Temporal Progression for Slanted Edge Model, 15° Yaw, 60, 100 and 140 km/h
Figure 5.36: OASPL, Frontal and Surface View, Steady State Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.37: OASPL, Surface View, Transient Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.38: OASPL, Top View (Base), Transient Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.39: OASPL, Top View (Bottom), Transient Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.40: OASPL, Surface and Top View, Steady State Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.41: OASPL, Surface View (Leeward), Transient Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.42: OASPL, Top View (Base), Transient Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.43: OASPL, Top View (Bottom), Transient Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.44: OASPL, Frontal and Surface View, Steady State Condition
Figure 5.45: OASPL, Surface View, Transient Condition, Semi Circular Model, 0° Yaw, 140 km/h

Figure 5.46: OASPL, Top View (Base), Transient Condition, Semi Circular Model, 0° Yaw, 140 km/h

Figure 5.47: OASPL, Top View (Bottom), Transient Condition, Semi Circular Model, 0° Yaw, 140 km/h

Figure 5.48: OASPL, Surface and Top View, Steady State Condition, Semi Circular Model, 15° Yaw, 140 km/h

Figure 5.49: OASPL, Surface View (Leeward), Transient Condition, Semi Circular Model, 15° Yaw, 140 km/h

Figure 5.50: OASPL, Top View (Base), Transient Condition, Semi Circular Model, 15° Yaw, 140 km/h

Figure 5.51: OASPL, Top View (Bottom), Transient Condition, Semi Circular Model, 15° Yaw, 140 km/h

Figure 5.52: OASPL, Surface and Frontal View, Steady State Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h

Figure 5.53: OASPL, Surface View, Transient Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h

Figure 5.54: OASPL, Top View (Base), Transient Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h

Figure 5.55: OASPL, Top View (Bottom), Transient Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h

Figure 5.56: OASPL, Surface and Top View, Steady State Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h

Figure 5.57: OASPL, Surface View (Leeward), Transient Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h

Figure 5.58: OASPL, Top View (Base), Transient Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h

Figure 5.59: OASPL, Top View (Bottom), Transient Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h

Figure 5.60: Comparison between Numerical and Experimental
Results of Peak Cp RMS for Reynolds Number Sensitivity, 0° Yaw

Figure 5.61: Comparison between Numerical and Experimental Results of Peak Cp RMS for Reynolds Number Sensitivity, -15° Yaw

Figure 5.62: Comparison between Numerical and Experimental Results of Peak Cp RMS for Reynolds Number Sensitivity, +15° Yaw
LIST OF TABLES

Table 1.1: Weighting Scale for Perceived Loudness                                               9
(adapted from Buley, 1997)
Table 2.1: Versions of the two-equation models                                                    55
Table 3.1: Boundary conditions, numerical schemes, turbulence and near wall model for FLUENT  121
Table 3.2: Boundary conditions, numerical schemes, turbulence and near wall model for AVL     122
Table 4.1: Percentage in Discrepancy of Results between GAMBIT and AVL Fame Hybrid Grid Generation Methods against Experimental Results 142
Table 4.2: Percentage Error Deviation of Models against Results of Alam (2000) at Various Yaw Angles 213
Table 4.3: Circular Models Vortex Size at 40% Scale                                              214
Table 4.4: RE Model Vortex Size at 40% Scale                                                    215
Table 4.5: SL Model Vortex Size at 40% Scale                                                    215
Table 4.6: Model Vortex Size Increase with Respect to the Horizontal Plane 216
Table 5.1: PSD and Frequency Peak for CAA                                                      294
Table 5.2: PSD Peak and Overall Discrepancy between CAA and Experimental                        296
Table 5.3: Transient Progression of Aero-Acoustics behind A-pillar Region 298
Table 5.4: OASPL of Vehicle Surface during Initial Transient State                             300
Table 5.5: OASPL Increase from Transient to Steady on Vehicle Surface                         301
Table 5.6: OASPL on Vehicle Surface during Steady State                                        301
Table 5.7: OASPL Reductions between Vehicle Surface and Domain during Steady State              303
Table 5.8: OASPL of Vortex Propagation on Vehicle Surface at Steady State Condition             303
Table 5.9: OASPL Reductions between Vehicle Surface and Domain during Steady State at Vortex Propagation Area
NOMENCLATURE

$\rho$ - Density

$u, v, w$ - Instantaneous Velocity in the x, y and z Component

$l, L, \ell$ - Length Scale

$\mu$ - Dynamic Viscosity

$I_u, I_T$ - Turbulent Intensity

$k$ - Turbulent Kinetic Energy

$U, V, W$ - Mean Velocity in the x, y and z Component

$\Omega$ - Vorticity Term

$x, y, z$ - Spatial Dimension in the Streamwise, Crosswise and Vertical Component

$p$ - Instantaneous Pressure

$P$ - Mean Pressure

$f$ - Frequency

$t_{ij}$ - Lighthill Stress Tensor

$S$ - Mean Strain Rate, Source Term

$\delta$ - Kronecker Delta

$t, T$ - Time

$\Delta$ - Del

$\phi$ - Transport Parameter

$\Gamma$ - Diffusion Coefficient

$\kappa$ - Karman Constant

$\tau$ - Turbulent Shear Stress

$\varepsilon$ - Dissipation Rate

$\omega$ - Specific Dissipation Rate

$u_c$ - Velocity Friction

$u^+$ - Dimensionless Velocity from the Wall

$y^+$ - Dimensionless Distance from the Wall

$\alpha$ - Coefficient of Proportionality
\[ I \] - Sound Intensity
\[ r \] - Radius
\[ \Pi \] - Cole Wake Strength Parameter
LIST OF ABBREVIATIONS AND ACRONYMS

BR – Bottom Row Pressure Tapings
CAA – Computational Aero-Acoustics
CAD – Computer Aided Design
CFD – Computational Fluid Dynamics
Cp – Coefficient of Static Pressure
DNS – Direct Numerical Simulation
FFT – Fast Fourier Transform
LE – Large Ellipsoidal Model
LEE – Linear Euler Equation
LES – Large Eddy Simulation
PISO – Pressure-Implicit with Splitting Operators
RANS – Reynolds Averaging Navier Stokes
Re – Reynolds Number
RE – Rectangular Edge Model
RMS – Root Mean Square
RNG – Re-Normalization Group
SE – Small Ellipsoidal Model
Semi – Semi-Circular Model
SIMPLE – Semi-Implicit Method for Pressure-Linked Equations
SIMPLEC – SIMPLE Consistent
SIMPLER – SIMPLE Revised
SL – Slanted Edge Model
SPL – Sound Pressure Level
St – Strouhal Number
TDMA – Tridiagonal-Matrix Algorithm
TR – Top Row Pressure Tapings
Chapter One
INTRODUCTION & LITERATURE REVIEW

In this chapter, a background introduction on vehicle aerodynamics, aeroacoustics and areas associated with it are presented. This is followed by relevant literature review that is relevant to the PhD project. Motivation that leads to this project will be later discussed together with the proposed method and scope of the project. This chapter concludes with presentation of the main objectives of this project and the layout of this thesis.

1.1 History of Vehicle Aerodynamics: A General Background

Studies on aerodynamics have originated from aeronautics and marine applications, Hucho (1998). According to Barnard (1996) at the turn of World War Two, substantial progress on aircraft aerodynamics was obtained due to the amount of research and analysis being done. Study of vehicle aerodynamics first began to surface during the earlier part of the 20th century and has continued up until the present day. During the earlier part of the 20th century, vehicle aerodynamics study is associated with vehicle performance, Hucho (1998). Aerodynamicists during that time carried out vehicle aerodynamics research with an aim to produce vehicles that can achieve a high speed to power ratio. To achieve high vehicle performance, much of the attention focuses on lowering the vehicle drag coefficient ($C_d$), which accounted to about 75 to 80% of total motion resistance at 100 km/h, Hucho (1998). However, in the later part of the 20th century, during the oil crisis of 1973-1974, the focus on vehicle aerodynamics study shifted towards lowering the drag coefficient in order to produce vehicles with better fuel economy, Hucho (1998).
The trend shifted again in the early 1990’s especially in North America where a low fuel price coupled with the increased popularity of light trucks and sport-utility vehicles have (of which drag coefficient of around 0.45), have reduced the importance the need on research to reduce drag coefficient, George et al. (1997). Aerodynamicists then shifted their focus towards designing vehicle that provides maximum comfort to its occupants. Vehicle comfort consists of fine-tuning areas such as ventilation, heating, air conditioning and minimising wind noise inside the vehicle, Hucho (1998).

1.2 Airflow Around a Ground Vehicle

Analysis of flows around a ground vehicle however, presented a different problem. As oppose to a streamline body of an aircraft, ground vehicle exists as a bluff body. The streamline feature of an aircraft causes airflow around it to be nearly two-dimensional. This results in airflow around the aircraft to be fully attached over most of its surface, Barnard (1996). On ground vehicle, the flows are strongly turbulent and three dimensional with steep pressure gradients, Ahmed (1998). According to Alam (2000), ground vehicles operate in the surrounding ambient turbulent wind that almost constantly present. This is different for aircraft since they travel above the turbulent atmospheric boundary layer. Furthermore, road vehicles can also travel at various high yaw angles depending on the nature of cross wind. Traveling at various yaw angles causes increased separated flow on the leeward side of the vehicle, adding more complexity to the flow field.

Airflow movement around the vehicle starts from the front. According to Barnard (1996), the airflow movement will cause the development of boundary layer close to the vehicle wall surface. The boundary layer thickness will increase as the airflow movement progressed around the vehicle.

Barnard (1996) classified the boundary layer generation on the vehicle wall surface into two stages; laminar and turbulent. During the initial stage, boundary layer flow exists in a laminar form. Near the front edge of the vehicle, the laminar
effect will cause airflow to slide over each other. Minimum skin friction drag formed between layers of airflow with the vehicle wall surface will cause the outer air layer moving faster than the inner one. This will slow down the flow. The slowing effect spreads outwards and the boundary later gradually becomes thicker. According to Barnard (1996), on most ground vehicles, the laminar boundary layer does not extend for much more than about 30cm from the front. Further downstream to the flow, instability develops and a transition to a turbulent flow takes place. In the turbulent boundary layer, the flow is still streamlined in the sense that it follows the contours of the body. The turbulent motions are still of very small scale. In the turbulent boundary layer, eddies are formed (groups of air molecules) resulting in rapid mixing of fast and slow moving masses of air (turbulent diffusion). The turbulent mixing will then move further outwards from the surface. However, very close to the surface within a turbulent boundary layer flow, a thin sub layer of laminar flow still exists. The two distinct differences between the flow mechanisms in the laminar and turbulent flow is that in laminar flow, the influence of the surface is transmitted outward mainly by a process of molecular impacts, whereas in the turbulent flow the influence is spread by turbulent mixing.

In the turbulent boundary layer, some of the energy is dissipated in friction, slowing airflow velocity, resulting in a pressure increase. If the increase in pressure is gradual, the process of turbulent mixing will cause a transfer of energy from the fast moving eddies in the turbulent boundary layer. If the rate of change in pressure is too great, for example in sharp corners, the mixing process will be too slow to push the slower air molecules moving. When this happens, the boundary layer flow stops following the contours of the surface, resulting in separation. Air particles downstream of the separation region will then move towards the lower pressure region in the reverse direction to the main flow. This is known as an adverse pressure gradient. Downstream of the flow, the separation region will reattach. The point between the region of separation and reattachment, where air is circulating is called the ‘separation bubble’. Separation will normally occur if the resultant flow encounters a sharp edge. It is always important for ground vehicles to have smoothly rounded edges everywhere. Each type of
separation can form a separation bubble zone either by reattaching itself downstream to the flow or it can be transformed into a wake, which recirculate frequently. Hucho named this frequent circulation as “dead water” zone, a term used in naval architecture. Farabee (1986) examined that the length of the separation bubble can be up to 100 times its height. Separation bubble zone happens normally on area in front of the windshield and on the side of the fenders while “dead water” zone normally happens on the rear surface of the ground vehicle.

The effect of separation and reattachment dominates most of the ground vehicle surface region. According to Ahmed (1998), vehicle aerodynamics operates mainly in the Reynolds number region in excess of $10^6$. Typical areas around the vehicle that exhibit small region of separation are the body appendages such as the mirrors, headlights, windshield wipers, door handles and windshield junction. Larger flow separation regions around the vehicle include the A-pillar\(^1\), body underside, rear body of the vehicle and in the wheel wells, Hucho (1998). (Refer Figure 1.1).

\[\text{Figure 1.1. Areas of Separation around a Vehicle (after Hucho, 1998)}\]

\(^{1}\) A-pillar of a vehicle is located between the front windshield and front passenger side door. The A-pillar base holds the side rear view mirror of the vehicle.
In a similar perspective, Ahmed (1998) defined the airflow as three dimensional with steep pressure gradients and having regions of separated flow. Regions of separated flow are categorized into small and large regions. Small regions of separated flow occur normally around attached component on a vehicle body such as headlights, mirror, door handles and windshield wipers. Large regions of separated flow occur on the A-pillar, at the rear of the vehicle, underneath the vehicle and around the wheel region.

Figure 1.2. Slanted A-pillar Vortex flow (after Hucho, 1998)

Although airflow around a ground vehicle exists predominantly in three-dimensional form, Hucho (1998) indicated that a quasi two-dimensional flow types also exist. The quasi two-dimensional type flow separates on the edge running perpendicular to the local direction of flow. The separation causes vortices to roll up with their axes almost parallel to the separation line. Turbulent mixing dissipates most of their kinetic energy making their development as continuing free trailing vortices, often weak and even untraceable. The quasi two-dimensional flow often occurs around areas such as the hood front edge and the front part of the vehicle windshield. Furthermore, according to Hucho (1998), the second type of separation normally separated at edges around which air flows at some angle. According to Hucho, the air stream then forms a cone-shaped helical vortex. The regions where these vortices most tend to be generated on a car are
behind the A and C pillars (Refer Figures 1.2 and 1.1). The axes of these vortices run essentially in the stream wise direction. The three dimensional vortices are very rich in kinetic energy and this containment in kinetic energy are determined by the ground vehicle geometrical conditions, mainly by the inclination of the A or C pillar angle at which they separate.

1.3 Overview on Sound and Noise

According to Barnard (1996), noise is the effect of pressure wave fluctuations transmitted to the human ear at the speed of sound. Sound waves reaches the human ear will travel through the ear passage to the eardrum causing it to vibrate. The three bones situated in middle ear region will then transmit and amplify the vibrations from the eardrum to the oval window in the inner ear region. Fluid in the inner ear then stimulates nerve endings, which will in turn send electrical impulses along the auditory nerve to the brain.

Sound pressure waves that are continuously received by the eardrum over time are random in nature (Figure 1.3). According to Buley (1997), the human ear can withstand over a large range of sound pressure variation that are being transmitted to it. The weakest sound pressure variations detectable by the human ear are of 20 μPa and because of this, a logarithmic scale was devised in order to provide a good subjective for the human ear to perceive loudness. The sound pressure level scale is measure in decibel (dB) and is defined as:

$$SPL = 10 \log \left( \frac{P}{P_0} \right)^2$$  \hspace{1cm} (1.1)

$P$ is defined as the root mean square of the sound pressure level with $P_0$ as the reference sound pressure level at 20 μPa.

According to Buley (1997), loudness is how the human ear perceive the intensity or energy of the sound pressure variation.
In the field of acoustics, the number of oscillation cycle it takes for sound pressure wave to complete in one second (or the number of periodic wave cycle completed in a given time period) is defined as Frequency, and is measured in Hertz (unit of $s^{-1}$).

$$Frequency = \frac{1}{T}$$ \hspace{1cm} (1.2)

In order to identify prominent frequency range in a random sound pressure wave distribution, Fourier analysis is conducted where a finite set of random sound pressure signal from a source is feed through a spectrum analyzer via a receiver (microphone), which then performs a Fast Fourier Transform (FFT). In a FFT, the random sound pressure signal is decomposed into several frequency bands, determined by band pass filters used in the spectrum analyzer. The end results are presented in a power spectra curve, often in a graph plot of SPL versus frequency, Callister et al. (1998).

According to Buley (1997), the human hearing ranges from 20 Hz to 20 kHz in frequency. The human ear is unique in such that its response to different types of frequencies are not equally sensitive. It is most sensitive at frequencies between
the ranges of 1 to 5 kHz and is not particularly sensitive frequencies that are either very high or low.

Fletcher and Munson (1933) investigated human hearing variation with various frequency and found that the ears respond to sound frequency differently at different perceived levels of loudness. From their findings, they then plotted a set of 'equal loudness' contours that can provide a way to measure loudness across a broad frequency range (Refer Figure 1.4).

![Figure 1.4. Fletcher Munsen Curve (from Buley, 1997)](image)

The loudness contours are measured in phon. For an example, at a 40-phon loudness level, a 63 Hz frequency must have a sound pressure level of around 58 dB to be as equally loud as a 1 kHz frequency with a sound pressure level of only 40 dB.

Because the human ear does not respond well against certain frequencies, a sound weighting scale was developed in accordance to how the human ear perceives loudness at different frequencies. There are four weighting scales (A, B, C and D) currently accepted and used worldwide for sound measurements. The ‘A’ weighting scale represents equal loudness at the 40-phon loudness level, Norsonic
It is normally used to measure sound in the 20 - 55 dB range, Harris (www.termpro.com). The ‘A’ weighting scale response similarly to the human ear in that it discriminates low frequencies and response effectively towards frequency in the 1 to 5 kHz range. The ‘B’ weighting scale represents equal loudness level at 70-phon and used to measure sound in the 55 to 85 dB range, Norsonic (www.norsonic.com), Harris (www.termpro.com). The ‘C’ weighting scale represents equal loudness level at 100-phon and used to measure sound above the 85 dB range, Norsonic (www.norsonic.com), Harris, (www.termpro.com). The ‘C’ weighting scale are normally used to detect sound pressure levels in the low frequency range. According to Alam (2000), the ‘D’ weighting scale is originally developed to evaluate aircraft noise measurements.

<table>
<thead>
<tr>
<th>Frequency Hz</th>
<th>32</th>
<th>63</th>
<th>125</th>
<th>250</th>
<th>500</th>
<th>1000</th>
<th>2000</th>
<th>4000</th>
<th>8000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Curve A dB</td>
<td>-39</td>
<td>-26</td>
<td>-16</td>
<td>-9</td>
<td>-3</td>
<td>0</td>
<td>+1</td>
<td>+1</td>
<td>-1</td>
</tr>
<tr>
<td>Curve B dB</td>
<td>-17</td>
<td>-9</td>
<td>-4</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-1</td>
<td>-3</td>
</tr>
<tr>
<td>Curve C dB</td>
<td>-3</td>
<td>-1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-1</td>
<td>-3</td>
</tr>
</tbody>
</table>

From Table 1.1, it can be seen that at 1 kHz (the lower threshold of human hearing sensitivity), there will be no adjustment needed for the A-weighting scale. Any sound pressure level located in a lower or higher frequency band will then be adjusted accordingly based on the perceived loudness of the human ear.

Based on the definition of noise provided earlier in this section, it can be said in general that noise is sound that is perceived as annoying. However, this remains a subjective matter since sound that is annoying to a person might not be annoying to another. According to Buley (1997), health, safety and environment are the three criteria that are used to assess the acceptability of noise. The Victorian Health and safety Regulations impose a limit to noise exposure for a maximum of 85 dB (A) in an 8 hours a day working environment. Any exposure higher than the limit imposed can cause hearing impairment to the employee. Several different of noise assessment scale are used to assess noise. The noise rating curve and the noise dose calculation chart are some of the example of the noise scaling system.
used. For motor vehicle in Australia, existing design rules (ADR28) allow a maximum limit of 90 dB (A) for a passenger car that remains stationary, with a maximum limit of 77 dB (A) while the car is in motion. In Europe, the rule is slightly stricter with 74 dB (A) as the maximum allowable noise limit for a passenger car to operate on the road.

1.4 Problems associated with Vehicle Vortex Flow

In most cases, turbulence is assumed as isotropic. Turbulent intensity can be then be written as:

$$T_l \equiv 100 \sqrt{\frac{2k}{3U}}, k \equiv \frac{3}{2} \frac{u^2}{u^2}, \frac{v^2}{u^2} \approx \frac{w^2}{u^2}$$

(1.3)

Where \( k \) is defined as the turbulent kinetic energy. \( T_l \) is given in percent.

In reality turbulent is always non-isotropic (three-dimensional). Strong sideways or cross-stream components of velocity on the surface of a ground vehicle complicate the formation and behaviour of the boundary layer. According to Barnard (1996), cross-stream components are more inclined to cause early transition of the turbulent boundary layer. Cross-stream flows can also keep the boundary layer attached by reducing high-pressure flow, making the pressure gradient less adverse.

A strong outward cross-flow can occur towards the edges of the windshield, producing separated vortices around the A-pillar region. These vortices are sources of both drag and aerodynamic noise. To curb their formation, it is necessary to ensure a smooth curve on the A-pillar and the windshield. In a vortex, the airflow velocity reduces with distance from the centre of the vortex. According to Roberson et al. (1997), vortex is defined as twice the average rate of rotation and can be written in its three-dimensional vector form as form as:
\[ \Omega = \left( \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right) i + \left( \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right) j + \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) k \]  \hspace{1cm} (1.4)

Furthermore, according to Roberson and Crowe (1997), vortex can be identified either as forced vortex or a potential vortex. In a forced vortex, the airflow velocity increases linearly from the vortex centre. In a free vortex, the airflow velocity decreases exponentially from the vortex centre (Refer equation 1.5). Forced vortex occurs due to the presence of viscous slipping between adjacent layers of fluid molecules.

\[ V_{\text{Forced}} = kr \]
\[ V_{\text{Free}} \propto \frac{1}{r} \]  \hspace{1cm} (1.5)

Real life vortex flow has a combination of both free and forced vortex structure (Refer Figure 1.5). The airflow velocity is high in the centre of the vortex, resulting in the formation of region of high negative pressure.

Figure 1.5. Forced and Free Vortex Velocity Distribution (from Roberson and Crowe, 1997)

Both quasi two-dimensional and three-dimensional vortices can lead to a development of skin friction and pressure drag on the ground vehicle. The total
drag production from the development of skin friction and pressure drag will result in the loss of performance and an increase in the vehicle’s fuel consumption. The main contributor of vehicle drag is the rear portion of the vehicle, which is not the focus of this study.

Apart from producing drag, the three-dimensional vortices are more detrimental in a sense that they also impose effects on the vehicles occupants. The vortices on the A-pillars impart stress on the front side windows of the ground vehicle. This will lead to the development of aerodynamic noise (also known as Aeroacoustics). Aerodynamic noise is then transferred to the passenger cabin that can be annoying and can cause both fatigue and discomfort to the occupants in the car after a long trip. Furthermore, interior vehicle noise makes it hard for vehicle occupants to communicate with each another and to listen to the radio or compact disc player. However, aerodynamic noise is a problem to the occupants at vehicle cruising speed of higher than 100 km/h (60 mph). At lower speed, the dominant noise sources are from the engine and tyres, George (1990), Callister et al. (1998).

According to Callister et al. (1998), the vehicle A-pillar area is a major wind noise contributor and efforts has to be taken in designing the A-pillar to reduce aerodynamic noise. Other vehicle body parts that are responsible for aerodynamic noise are the junction between the bonnet and windshield, the roof racks, the vehicle C-pillar, and gaps between the doors. In addition, add on parts on the vehicle, such as the radio antenna, windshield wipers and external rear view mirrors are also contributors to aerodynamic noise. George (1990) defined these add on parts as parasitic noise source. The A-pillar is relatively close to the front seat occupant’s ear, so noise in this area is noticed readily. The flow around most A-pillar is separated, causes intense turbulent vortex flow to form on the side window behind the A-pillar. Noise generated by the A-pillar is of broadband type in nature and in the low frequency region, caused by the large scale turbulent eddies from the A-pillar vortex separation, Haruna et al. (1990, 1992), George et al. (1997), AVL (wwwavl.com), (2003). There are several reasons why the A-pillar contributes highly towards interior noise generation. One reason is that a few vehicle body components are joined together around the A-pillar area.
(windshield, the door, the outside rear view mirror and the front vehicle quarter panel). Problems that usually suffice from this are normally due to poor sealing and fitting problems. Callister (1998) explained that an auxiliary seal is definitely needed to seal the A-pillar gap on doors with fully framed windows. This will stop the pressure fluctuations to creep inside the vehicle transferring unwanted noise. Another reason for noise generation around the A-pillar area is due to the fact that flow around the A-pillar possesses relatively high velocities. Any exposed cavity or protuberances will cause a high level of wind noise. Callister (1998) quoted Watanabe et al. (1978) in saying that the flow around the A-pillar is normally at around 60% higher than nominal free stream velocity. Considering that wind noise starts to impose problem at speed above 100 km/h, close to the surface of the car and around the A-pillar, the air velocity will be around 160 km/h. This increase in local velocity will result in a low local pressure level, especially at the core of the vortex, Barnard (1996). The resulting dipole type, high sound pressure level in the low frequency region (100 to 500 Hz) is proportional to the sixth power of velocity, Haruna (1992). In accordance to the sixth power proportionality rule, George (1990) indicated that local area of separation with coefficient of pressure ($C_p$) of $-1.0$ will result in a 9dB sound pressure level increase while a $C_p$ value of $-2.0$ will result in a sound pressure level increase of 14dB. In addition, any wind noise located around the A-pillar region is around 17 dB louder than a source exposed to the free-stream velocity, Callister et al. (1998).

1.5 Vehicle Noise

According to Ahmed (1998), there are two types of noises that are of a concern to vehicle designers. The noises are drive-by noise, which affects people outside the vehicle and interior noise, which affects the driver and passenger. George et al. (1997) described that the vehicle noise heard by people outside the vehicle is called exterior vehicle noise and the vehicle noise heard by the automobile occupants as interior vehicle noise.
Exterior and interior vehicle noise is transferred to the surroundings either through the vehicle structure (structure borne) or via external airflow around the vehicle (air borne). Structure borne noise originates through vibration of vehicle structure. An example of structure borne noise is noise that originates from vehicle tire dynamic interactions with the road surface. Another example of structure borne noise is vibration effects from vehicle mechanical components such as vehicle powertrain systems (engine and transmission). Air borne noise originates through forces generated from air that flows around and through the vehicle. Examples of air borne noise are noise generated from the vehicle ventilation and exhaust system, engine air intake, A-pillar and side mirror.

Exterior noise originates mainly from the powertrain systems and tyres. However, according to George (1995), extensive efforts have been put forward over the years to minimise engine and tyre noise. Work has been done towards reducing engine noise such as adding sound absorbent materials material surrounding the engine compartment and developing larger capacity mufflers to reduce exhaust noise. Throughout the years, manufacturers have succeeded in reducing tyre noise. According to Affenzeller et al. (2003), for modern day tyres, emphasis have been put on modifying tyre tread, making them more randomised to avoid high tonal noise and making the grooves on tyre tread more ventilated for better pressure distribution. This is attributed to the fact that modern tyres are wide and offers better grip on the road surface, thus making it much noisier. For vehicle speed below 100mph (160 km/h), reduction on tyre noise has been lower than engine and drive train noise, George (1990).

Interior vehicle noise originates from various sources from the vehicle. Vehicle ventilation system, engine and tires are contributor to vehicle interior noise. Engine and tyre noise contributes to interior noise predominantly at low vehicle speed. Interaction between air and the external vehicle body parts also contributes to interior noise. This phenomenon is classified as ‘aerodynamically induced noise’ or ‘aerodynamic noise’ and is the domain of vehicle aeroacoustics study. According to George (1990, 1995) aerodynamic noise starts to become dominant when vehicle is traveling at high speed, at around 70 mph or greater. Furthermore,
George (1995) stated that aerodynamic noise is dominant in frequency region of between 500 to 12 kHz. At present, aerodynamic noise is seen mainly as problem of the internal environment rather than the external environment of the vehicle. According to Callister et al. (1998), interior vehicle noise is annoying because it makes it harder for occupants to communicate with each another. Furthermore, it makes it hard to listen to the radio or compact disc player. Moreover, interior noise can cause fatigue to the driver on long trips. According to George (1990) interior noise is causing significant comfort problems at cruising speed of around 60 mph (96 km/h) and above. Interior noise around this vehicle speed ranges between 70 to 80 dB, making long trips inside a vehicle discomforting and tiring to occupants. Barnard (1996) reviewed a study on a small car at 150 km/h and found that the engine contributes to 82.5 dB of interior vehicle noise. Tires and aerodynamic noise contributes to 78.0 dB and 78.5 dB respectively. Barnard (1996) quoted Buchheim et al. (1968), which conducted a study on 15 different vehicles and found interior noise level at vehicle speed of 113 km/h ranges between 62 to 78 dB (A) and rising to 72 to 87 dB (A) at 180 km/h, which is slightly higher than the industrial workplace limitations of 85 dB (A).

In addition, a vehicle with low aerodynamic drag does not necessarily will have low levels of aerodynamic noise. Buchheim et al., surveyed fifteen production cars in 1982 and found that aerodynamic drag and aerodynamic noise are independent of each another. Aerodynamic drag depends predominantly on the exterior airflow over the rear of the car where flow separation is occurring while aerodynamically induced vehicle noise depends mainly on exterior airflow around the A-pillar and windshield where small openings or imperfectly seal of the doors and windows that may generate strong unsteady pressure fluctuations that resulting in vehicle interior noise generation, Callister et al. (1998). George et al. (1997) added that aerodynamic drag depends on the transient mean pressure distribution on the vehicle surface. However, vehicle aerodynamic noise depends on the strength of the surface pressure fluctuations relative to the mean value.
1.6 Mechanism of Aerodynamic Noise Generation

Callister et al. (1998) described that for aerodynamic noise, the generation mechanism must include a ‘source’, ‘path’ and ‘receiver’. The ‘source’ is described as the area where energy is converted into acoustic energy. The acoustic energy then radiates from the source location and is transmitted through different mediums i.e. liquid or through solids. George et al. (1997) described the interior vehicle aerodynamic noise ‘source’ as the fluctuating pressure caused by the turbulent flow around the car, flow over gaps and protrusions and leaks. The ‘path’ is described as the route along which the acoustic energy is transmitted on its way to the receiver. It can be either through clear travel passage through leaks or cavity or through vibration of the vehicle body shell radiating acoustic energy into the vehicle. The ‘receiver’ is the person or microphone that receives the acoustic energy and converting it into sound pressure signals.

Identification of aerodynamics noise source can be done through the development of idealized models. According to Callister et al. (1998), aerodynamic noise can be classified into either monopole, dipole and quadrupole idealised model.

The monopole source effect originates from an unsteady introduction of volume into the surrounding fluid. It is the most efficient sound generator at low mach numbers. The most notable monopole source of noise for automobiles comes from unsteady volumetric flow addition. If a fluctuating pressure on the vehicle exterior surface causes an unsteady volumetric flow addition to the interior of the car through a leak path, then a strong secondary monopole sound source will result. Callister et al. (1998) quoted Norton (1989), in stating that a good example of monopole sound is to come from the un-muffled vehicles engine intake and exhaust pipe.

Dipole source effect is the next most efficient generator of sound at low Mach numbers. Dipole effect is the caused by unsteady forces to the fluid resulting in unsteady pressures to act upon rigid surfaces on a vehicle. Noise from a separated turbulent flow impinging upon a surface is an example of dipole noise.
Automobiles typically have numerous separated flow regions with the A-pillar being arguably the most popular with its aerodynamic noise generation capabilities.

The least efficient sound source at low Mach numbers is called the quadrupole source effect. Quadrupole source effect is caused by internal stresses and turbulence within the flow. It is best described as two fluid elements colliding with each other, as might happen in a turbulent shear layer. Jet noise is an example of quadrupole source effect Goldstein (1976). Quadrupole source effect can usually be ignored in automotive flows since they are comparatively very weak when compared to monopole or dipole type source effect.

George et al. (1997) identify the intensity of monopole, dipole and quadrupole acoustic models as:

\[
I_{\text{monopole}} \sim \frac{\rho}{r^2 c} L^2 V^4 \\
I_{\text{dipole}} \sim \frac{\rho}{r^2 c^3} L^2 V^6 \\
I_{\text{quadrupole}} \sim \frac{\rho}{r^2 c^5} L^2 V^8
\]  

(1.6)

The sound intensity produced by a monopole, dipole and quadrupole source can be seen from equation 1.6 to be proportional to the flow velocity raised to the fourth, sixth and eighth power respectively. The formulas also show that by dividing the source effect intensities to find the ratio of their strengths, it can be seen that the dipole source strength divided by the monopole source strength is proportional to the Mach number squared. The quadrupole source strength divided by the dipole source strength is proportional to Mach number squared. This shows that at low mach numbers, if there is a monopole source, it will be the primary noise source. When no monopole source is present, any extant dipole sources will be dominant. Similarly, at low Mach numbers the quadrupole effects are important only if both monopole and dipole effects are negligible.
Automobile aerodynamic noise is typically a mixture of monopole and dipole sources. According to George (1990), automobile aerodynamic noise is proportional to the sixth power of the flow velocity. This was confirmed by a study conducted by Haruna et al. (1992). Therefore, this explains the reason vehicle aerodynamic noise dominates tire noise and engines noise at high vehicle speeds.

Earlier in this sub-section, the concept of source, path and receiver was mentioned for the complete transmission of aerodynamic noise to take place. However, this is a generalized concept for aerodynamic noise generation. In reality, the aerodynamic noise can be transmitted either through a leak, cavity or can be generated through airflow turbulence interaction with the vehicle body.

Aerodynamic noise generated through a leak is called a leak noise (or sometimes called aspiration noise) and it can occur in two ways. According to Callister et al. (1998), leak noise could be caused by movement of airflow through an area of small leaks, which connects the exterior, and the interior of the vehicle. George (1990) further added that leak noise propagation could also be transmitted through panels, windows and seal, due to the fact that their transmission losses are less than 100%. Both mechanism of leak noise can be seen in Figure 1.6.

Figure 1.6. Two method of Leak Noise Transmission (From Callister et al., 1998)
Callister et al. (1998) further described that airflow movement at a rather high velocity causes leak noise, transmitted from a high-pressure zone to a lower pressure zone. George et al. (1997) further mentions that it is not unusual for leak noise to increase interior noise by as much as 10 dB. Leaks are normally present on vehicle door seals, movable glass seals and the fixed glass seals. Furthermore, according to George et al. (1997, leak noise can be transmitted by either through steady or unsteady pressure from the vehicle exterior. If the leak comes into contact with the external surface of the vehicle that is experiencing turbulence separation and generating pressure fluctuation, then the mass flow entering the leak will be unsteady, generating monopole sound inside the car. The tonal and fluctuating nature of monopole sound will result in a high frequency noise, which is noticeable and annoying, George et al. (1997). This is illustrated in the top part of Figure 1.6 where a defective seal can generate monopole sound due to fluctuating external pressure at location one. This will affect the mass flow through to location two. Secondly, if the leak connects to a steady pressure source generate steady flow of air through the leak opening, flow will only starts to separate in a turbulent manner after the location two areas, thus generating local fluctuating pressures. This gives rise to dipole type sound, which is transmitted into the vehicle.

The second mechanism of leak noise can be seen in the bottom part of Figure 1.6, which shows how leak noise can be transmitted through a seal, even when there are no leaks. According to George et al. (1997), the external pressure at point one can move the seal forward and backwards slightly and generate sound. The seal will absorb some of the noise. According to George et al. (1997), by doubling the seal mass, sound pressure level will increase only by 3 dB in attenuation. Using multiple seals however, can provide up to 5 dB in interior noise reduction in some cases. George et al. (1997) referred to the work by Danforth et al. (1996) for a recent experimental study on effect on single and double seal on noise flow. Other references on leak noise mechanism can be seen from Callister et al. (1998), who referenced the work of Jung et al. (1995), describing a recent study by the authors on the influence of leaks on the interior wind noise level.
The second method of aerodynamic noise transmission is through a cavity, and can be described as cavity noise. As per leak noise, cavity type noise is also often located in region of high velocity flow, such as the exposed gaps around the A-pillar area or around the outside rearview mirror, Callister et al. (1998). George et al. (1997) divides cavity noise into two categories namely large (i.e. open windows and sunroofs) cavities and small cavities (i.e. door gaps) respectively. George (1990) also divided cavity noise into two types, broadband type noise and tonal type noise. Similar to leak noise, cavity noise is of monopole and dipole type origin.

George (1990) and Callister et al. (1998) described broadband noise as noise caused by turbulent boundary layer flowing across the cavity creating trailing edge noise as it passes over the cavity. A turbulent free shear layer then develops as the flow leaves the cavity. It then impinges itself on the rear of the cavity generating leading edge noise, resulting in cavity noise displaying broadband frequency characteristics (Refer Figure 1.7).

George (1990) and Callister et al. (1998) further described that besides the broadband cavity noise, a tonal type cavity noise can also be generated. Tonal type cavity noise generation involves a feedback and resonance mechanism. Similar to broadband type cavity noise, the tonal type cavity noise involves disturbance shedding from the front edge of the cavity. This disturbance impinges on the rear edge of the cavity generating acoustic wave tones that propagates in all directions. When the acoustic wave reaches back to the front edge of the cavity, it
then triggers another shedding of disturbance, giving it a feedback and resonance type phenomenon. According to George (1995), this feedback can be acoustic or convective and it can involve ordinary acoustic modes of a cavity or Helmholtz type resonance (Refer Figure 1.7).

George et al. (1997) further adds that large cavity noise generates mostly low throbbing frequency noise, which can be both annoying and fatiguing. Small cavity noise on the other hand is most likely to generate high frequency noise. High frequency noise is much easy to absorb with carpeting and upholstery. Low frequency noise however, is more difficult to absorb. Helmholtz type resonator damping, damped panels or active damping will have to be used in order to minimize it (George, 1990). Callister et al. (1998) referred the work Rockwell et al. (1978) on cavity noise for further reading.

The third and final method of aerodynamic noise mechanism is due to turbulence airflow interaction with the vehicle body. This can be described as wind rush noise, Callister et al. (1998). It is generated by fluctuating pressures on the exterior of the vehicle caused by the fluctuating, unsteady turbulent airflow over the surface (Refer Figure 1.8). Wind rush noise will always be present over the vehicle surface even though the vehicle surfaces were perfectly rigid and leak-free and the flow are attached throughout the vehicle surface due to the turbulent nature of boundary layer on the surface of the vehicle. However, if the flow is separated, the noise generated will be intensified by a factor of approximately ten, Callister et al. (1998). Wind rush noise will initiate dipole effect source type effect, radiating outward in all directions and since the vehicle is not perfectly rigid, the fluctuating pressure impinging on the vehicle windows and body panels will result in vibration, radiating noise into the vehicle interior. In addition, wind rush noise is broadband in nature but is less annoying than tonal noise.
1.7 Ways of Reducing A-pillar Aerodynamic Noise

In vehicle design, the aim is to reduce interior noise through altering vehicle body design, without adding extra component to the vehicle, as this will add additional cost to the overall vehicle production. In order to reduce aerodynamic noise generation at the A-pillar region, the size and intensity of the A-pillar vortex must be reduced. In theory, Haruna et al. (1990) proposed that aerodynamic noise is dependent on properties such as the surface fluctuating pressure, the frequency and the correlation area. Therefore, minimisation of aerodynamic noise could be done by suppressing the surface fluctuating pressure and frequency, and by reducing the correlation area. According to George (1990), high frequency noise that propagates into the vehicle can easily be absorbed by carpet and upholstery. Low frequency noise can be reduced through the use of Helmholtz resonator damping, damped panels or active damping.

In a physical sense, one way to reduce aerodynamic noise is to design vehicles with a small inclination angle between the windshield and bonnet, Scibor-Rylski (1984). Alam (2000) mentioned in his work that the average windshield inclination angle for a passenger car is around 60°. However, it was mentioned by Hucho (1998) that small windshield inclination angle with the bonnet will not further reduce drag and it also impose visibility and temperature problems to the
vehicle occupants. Another way of reducing A-pillar vortex size and intensity was recommended by Callister et al. (1998) in that the radius of the A-pillar has to be large to further minimise the airflow velocity and turbulence intensity. This proved to be true. Experiments conducted by Alam (2000) on simplified vehicle models at various yaw angles\(^3\) have showed that by increasing the A-pillar windshield radius, a decrease in ‘in cabin noise’ was obtained particularly in the leeward side of the vehicle. It was also recommended by Callister et al. (1998) that the A-pillar region to be designed without exposed rain gutter. Exposed rain gutters will usually cause wind noise as it cause flow to separate at the edge of the A-pillar. Again this proved to be true based on experiments carried by Alam (2000), which showed that by adding a rain gutter on the A-pillar, an increase in fluctuating pressure was obtained behind the A-pillar region. A study by Piatek et al. (1989) has shown a simple modification carried out on the vehicle A-pillar to redirect rainwater without having to design an exposed rain gutter.

According to Callister et al. (1998), outside rear view mirrors also exists as a bluff body that contributes to the disturbance of the flow pattern around the A-pillar area. Coupled together with the A-pillar vortex, airflow separation behind the A-pillar can result in high wind noise levels. A study made by Hamel et al. (1996), have showed that the rear view mirror contributes to the increase of wind noise up to 20 dB (130 dB with mirror and 110 dB without mirror) at region downstream close to the side mirror, dominating in the frequency spectrum range of lower than 1 kHz. Lokhande et al. (2003) conducted an LES simulation of a generic side view mirror and also managed to obtain a sound pressure level peak of around 130 dB behind the mirror region. In addition, Fukushima et al. (1995) conducted a study on aerodynamic noise from a side mirror and found out that pressure fluctuation level generated behind the rear view mirror are around 90 to 100 dB. Callister et al. (1998) recommended that the external rear view mirror to be moved as far rearward as possible in order to minimise wind noise. This moves the mirror out of the maximum flow speed area. However, careful placement must be done so that the field of view of the driver is not restricted. In addition to that,

\(^{3}\) To simulate the effect of crosswind, vehicle or scaled model is rotated to various angle in the wind tunnel. This varying angle of rotation is called the yaw angle.
the exterior overall shape of the mirror must be carefully designed to minimise induced wind noise. Normally aerodynamicists will design the rear view mirror first aiming in achieving low drag and only secondly to minimise wind noise, George (1997). Mirrors with a rounded housing are generally preferred with holes and gaps in the mirror housing to be eliminated. This is to minimise trailing edge noise generated by flow separation past sharp edges along the rear view mirror, Callister et al (1998), George et al. (1997).

1.8 Vehicle Aerodynamics and Aeroacoustics: Numerical and Computational Evaluation Methods

Numerical evaluation methods involving vehicle aerodynamics and aeroacoustics can be done either analytically or by using Computational Fluid Dynamics (CFD). Analytical methods in solving airflow behaviour realistically can be done on simple generic type flow problems in either two-dimensional or three-dimensional form. As airflow behaviour gets more complex when subjected to flow around complex geometrical domain or bluff bodies, (with the presence of turbulence or compressibility effect), solving airflow properties cannot be done analytically. This is because in order to obtain its complete turbulent and aerodynamic noise source properties, full unsteady Navier-Stokes (taking into account inertia, viscous and pressure forces) together with the continuity equation (mass conservation) need to be solved.

However obtaining a direct numerical solutions of Navier-Stokes equations are still not yet possible even for modern day computers. The main reason being that grid points needed for a typical CFD model to be solved are Re\(^{9/4}\). For a typical flow with Reynolds number of 10\(^6\), it will take the computer to generate and solve equations for 3.16x10\(^{13}\) grid points. This is far beyond the reach of even the most state of the art supercomputers available in the world today. In order to come up with a comparable solution, steady or time averaged Navier-Stokes equation is used (called Reynolds Average Navier-Stokes equation – RANS) together with turbulence model, developed to take into closure problems involving Reynolds
stresses resulting from the time averaging process. Solving for RANS, continuity and turbulence model equations can be done via Computational Fluid Dynamics (CFD) simulations.

CFD approach for turbulence modeling was first intended for the aerospace community in the 1960s and 1970s (Anderson, 1995). In the early development stage of CFD for automotive applications, codes were expected to provide actual quantitative data that is similar to measured wind tunnel data. Knowing that this is not yet possible, present use of CFD in automotive are used to provide information about flow characteristics and phenomena, which dictates aerodynamic performance. However, the ultimate goal in CFD is to obtain model as flow as actual as possible and current and future research on CFD is ongoing in order to achieve that goal. Furthermore, current applications of CFD in the automotive industry are determined by economics viability. To be economically viable, the codes should be able to simulate the correct physics of the flow and at the same time achieve computational turnaround time that is the same or less than that of a wind tunnel test cycle time. Ahmed (1998) has showed that for a typical vehicle, current testing time taken in a wind tunnel in order to achieve desired level of $C_d$ reduction has increased. This will be an expensive exercise for automobile manufacturers. With the reduction on computational cost, aerodynamic simulation by using CFD, being run at a faster turnaround time will only be at a fraction of the cost.

However, this will only put more demand on the current performance of computers speed and memory. These are due to several factors:

- An increase sophistication of flow physics modelled.
- An increase in modelled geometries complexities.
- An increasing number of multidisciplinary approaches of flow simulation.

These increases in computational demands are intended in achieving the ultimate goal ‘fluid flow realism’ in CFD simulation as mentioned earlier. A much more complex three-dimensional vehicle geometries are now being used in automotive
CFD simulation coupled with high grid density to achieve better flow resolution. Usually this also leads to a more accurate and realistic flow simulation. In an unsteady three-dimensional flow, a doubling of grid density (to double the accuracy) results in a (with three space coordinates and a time dimension) sixteen-fold increase in computation effort. In addition, better CFD post processing flow visualisation effect such as colour-coded pressure distributions over the entire body surface and observation of particle traces in real time animation also puts extra demands on computer speed and memory.

From the CFD results obtained through RANS simulations, aeroacoustics properties of the flow can be extracted and analyzed. This branch of CFD technique is called Computational Aeroacoustics Analysis (CAA). In CAA the surface fluctuating flow data obtained from either steady state or time dependent CFD simulation are used as source terms for CAA simulation. CAA simulation technique can be categorized into three, which are the Lighthill Acoustic Analogy Method, Kirchoff Method and Perturbation Method. Majority of study on CAA incorporate the Lighthill Acoustic Analogy to evaluate for aerodynamic noise propagation. The Lighthill Acoustic Analogy was made famous by the late Professor Sir James Lighthill where in 1951 and 1954 introduced papers proposing a theory on noise generation in free stream flow. Efforts were made by Curle (1955, Ffowcs-Williams and Hawkings (1969) to extend Lighthill’s work to include modeling of aerodynamic noise propagation around a solid body.

Computational Aeroacoustics Analysis is still new and is open to a lot of discussion and ideas. According to Ogawa et al. (1999) in a review on aerodynamic noise prediction using CFD, the accuracy of aerodynamic noise prediction is dependent on the computational accuracy in solving transient flow. However, similar to CFD, more work needs to be done with CAA especially on flow around the A-pillar region in order to evaluate its performance under various geometrical and flow environment. CAA based on the CFD results is limited to external near-field noise radiated close to the vehicle surface. Recent recommendations on how to better predict interior wind noise by combining data from CFD and Statistical Energy Analysis (SEA) is given by Bremner et al.
By using SEA, researcher is able to predict structure borne noise that’s being transmitted inside the car caused by external pressure fluctuations on the vehicle surface. Other numerical analysis that uses SEA to predict structureborne interior vehicle noise are given by De-Jong (1985), Yashiro et al. (1985), Strumolo (1997), Yang et al. (1997), Iida (1999) and Manning (2003).

1.9 Literature Reviews on A-Pillar Aerodynamics and Aeroacoustics

Over the years, research studies concerning A-pillar aerodynamics have focused mainly in understanding the mechanics of airflow behaviour when exposed to various A-pillar and windshield configurations to help further reduce aerodynamic noise. Research studies conducted are predominantly using experimental and numerical method or a combination of both.

Stapleford et al. (1970) conducted experimental studies of aerodynamic noise generation on a yawed rectangular shaped box at different yaw angles. External aerodynamic noise was measured from various region of airflow. From their studies, they found out that the aerodynamic noise generation was highest at the region of vortex flow behind the A-pillar region. Aerodynamic noise generation from vortex flow is around 120 dB, a 20 dB increase from the background tunnel ambient noise. They also found out the highest sound pressure level was in the region of low frequency, due to the large-scale turbulent structures at the area of flow separation.

Fricke in (1968) and (1971) conducted a study on pressure fluctuations on separated flow and concluded that the wall pressure fluctuations of subsonic separated flow are an order of magnitude higher than those beneath a boundary layer and that the source of wall pressure fluctuations is in the shear layer above the re-circulating flow, close to the reattachment point. This was different to the findings of Mohsen (1967) in which he discovered that maximum pressure fluctuations occur near the reattachment region of the flow.
Laufer (1974) explained that noise formed from vortex generation is due predominantly by vortex pairing or mixing. However, Hussain (1983, 1986) argued that not all the vorticity component play a role in vortex noise generation, with the breakdown of flow geometry i.e. separation, reattachment and vortex breakdown also playing an important role.

Watanabe et al. (1978) experimented with a slanted angle A-pillar model that showed a conical vortex structure generated behind the A-pillar region. High-pressure region centered at the vortex core with intense pressure distribution being strongest at the A-pillar base and area close to the A-pillar ($C_p$ values is around -2.0). The vortex flow grew weaker as it rotates further from the A-pillar base, particularly around the roof region, next to the A-pillar ($C_p$ values is around -0.7).

Buchheim et al. (1982) conducted experiments to investigate interior noise level due to external aerodynamic noise generation on various components of a vehicle. Around the A-pillar region, they obtained similar findings as per Stapleford et al. (1974) in that aerodynamic noise generation was 20 dB higher from the background noise. However, after modification of the A-pillar region, they managed to reduce the aerodynamic noise generation around the A-pillar region to the same level as the background noise. They concluded from their study that no particular component of the car dominated in radiating interior noise. However, interior noise level radiating from the A-pillar region is the highest at 60 dB (A) and corresponds to region of low frequency (250 – 500 Hz).

Simpson (1987, 1989) explained that the effective pressure fluctuations of vortex flow might be near the locus of maximum shear stress position of the separating turbulent boundary layer. The large-scale motions produced in the vortex flow separation do not contribute much to the turbulent shear stresses. It only changes the mean flow-field to produce low frequency pressure fluctuation at low Mach number.
Sadakata (1988) showed that at a critical A-pillar slant angle between 40° to 50°, a sudden increase in sound pressure occurred. Sadakata and his colleagues concluded that noise could be reduced considerably by combining the proper A-pillar slant angle with curved windshield and smooth rounded A-pillar geometry.

Bearman et al. (1989) conducted several experimental tests with numerical type validation of a passenger car and simplified scale models. The tests were carried out to examine the effect of vortices generation in vehicle under wind yaw condition while exposed to wind of 20m/s. A maximum yaw angle of 20° was investigated with increment of 5°. It was observed that the effect of vortex increases at higher yaw angle in the leeward region of the car with sudden escalation after 10°. It was not mentioned however in the paper of the windshield angle and radius used in the experiment. When surface pressure measurements were taken on the surface of the A-pillar region (did not mention where exactly) at 0° yaw, it was found that minimum Cp was between –1.4 and –1.5. In addition it was stated that the A-pillar region of flow were highly unsteady and the vortex strength kept changing with time.

Haruna et al. (1990) experimented using a production car in a wind tunnel at 50 km/h and yaw angles of 0° and 10° respectively. From their study, they found out that the separated region of the flow at 10° yaw is larger when compared to the separation region at 0° yaw. They also found out that high sound pressure level originated from side window surface fluctuations was sustained at a large area when the car was yawed at 10°. High sound pressure level was observed at region of low frequencies. The overall sound pressure level at when the vehicle is yawed was at its highest at around 110 dB (A) and a difference of 10 dB (A) was recorded when compared against vehicle at 0° yaw. Furthermore, they discovered a primary and secondary vortex rotating opposite each other behind the A-pillar. The vortex generated when the vehicle is at 10° yaw is greater in size (around 85-mm in cross section) in comparison to the vehicle at 0° yaw angle (around 50-mm in cross section). In addition, they also found that the separated region exhibit rotational flow with the reattachment region having irrotational flows. However,
they failed to measure the size of vortex progressively throughout the span wise length of the A-pillar.

Popat (1991) experimented on effects of windshield angle on A-pillar vortex. Popat noted some Reynolds number sensitivities at different A-pillar slant angle except for when the inclination angle is at 60°. Popat noted in his thesis three stages of vortex formation at different A-pillar slant angle with only bubble separation occurring below 20°, vortex-bubble separation occurring between 30° to 40° and fully developed conical vortex occurring between 50° to 60° slant A-pillar angle, which is what normally experienced on a normal production car. Popat concluded that the critical angle at which peak mean and fluctuating pressure values occurred at 40° inclination.

Haruna et al. (1992) developed a numerical model to estimate the distribution of surface pressure fluctuation for aerodynamic noise prediction. They also validated their model with experimental data. From the experimental data, they found out that the highest contributor of aerodynamic noise is the A-pillar with overall sound pressure level ranging between 110 to 130 dB, at different vehicle velocities of 50, 100 and 140 km/h respectively. The aerodynamic noise generated was dependent to the fourth power of vehicle velocity. From their experimental results, they found out that high surface pressure fluctuation occur at the base of the A-pillar region. Highest sound pressure level was predicted at 105.5 dB, which correspond to a frequency of 500 Hz. From their model, they obtained predicted aerodynamic noise distribution behind the A-pillar region at frequencies range of between 100 – 500 Hz for vehicle velocity at 50 km/h, 200 – 1 kHz for vehicle velocity at 100 and 300 – 1.5 kHz for vehicle velocity at 150 km/h. At 50 km/h, the highest sound pressure level was obtained at 107.7 dB at 150 Hz. At 100 km/h the highest sound pressure level was obtained at 120.6 dB at 300 Hz and at 150 km/h the highest sound pressure level was obtained at 127.0 dB at 450 Hz. In addition, noise intensities were also predicted. Highest recorded noise intensities were 58.9, 76.8 and 83.9 dB, which corresponds to vehicle velocities of 50, 100 and 140 km/h respectively. From their model, the aerodynamic noise was at fifth power to the vehicle velocity.
Haruna et al. (1992) developed a numerical model to analyze aerodynamic noise on a delta wing (to mimic A-pillar flow) for incompressible flow. Numerical investigation was conducted at six different velocities between 50 to 200 km/h. The numerical model that was developed was divided into two parts. The first part was developed using continuity and unsteady Navier-Stokes equation to solve for flow properties on the delta wing. However, Haruna failed to mention the time steps taken in the simulation. Results from the first part were then used to solve for aerodynamic noise. Equations used for the outer region was developed from the Lighthill - Curle’s for acoustic radiation from a solid body. Results of fluctuating pressures against time were obtained. Results did not show good agreement against empirical data although it showed similar trend. They also obtained spectral analysis of the aerodynamic noise prediction. Spectral analysis against measured results showed that the predicted results were under predicted by about 20 dB. However, predicted results managed to capture the trend of results obtained from measurements. The author concluded that the under prediction might be caused by insufficient amount of grids generation. High sound pressure level was obtained at frequency region of below 1 kHz. In addition, results showed sound radiated from the delta wing to be of dipole sound strength, which correspond to the power of sixth velocity, which is in agreement to the Lighthill-Curle’s equation, (Curle, 1955).

Hanaoka et al. (1993) conducted a numerical simulation to determine A-pillar aerodynamic noise behaviour at different windshield slant angle (30°, 45°, 52.5°, 60° and 75°s from horizontal axis) at 100 km/h using quasi DNS method (at time steps of 10 mili-seconds with 0.6 second sampling case to correspond at Courant number of 0.4) and Lighthill-Curle equation (to solve for noise). Results from the simulation showed an increase in wind noise with windshield angle especially at critical angle of above 50°. Results also showed that main aerodynamic is of dipole sound source and originated from area of vortex separation and reattachment together with turbulent shear interaction with the side window roof area. Furthermore, pressure spectra results showed high wind noise at low frequency range caused by large scale eddies. The highest peak of predicted
Overall sound pressure level was obtained at 110 dB for the 75° vehicle model and the lowest peak of predicted overall sound pressure level was obtained at 45 dB for the 30° vehicle model. In addition, time averages surface pressure distribution was also predicted. The highest $C_p$ value obtained was at the center of the vortex, which location shifted closer to the base of the A-pillar as the slant angle increases. Highest recorded $C_p$ value was $-4.0$ at 60° slant angle and the lowest recorded was $-1.6$ at 30° slant angle. However, study failed to show any comparison with empirical data and points of measurement for predicted overall sound pressure level.

Nienaltowska (1993) experimented with flow behind the A-pillar at various velocity and measured pressure and velocity fluctuations at points away and perpendicularly from the side window. Nienaltowska found that turbulence generation is independent of velocity and that it decreases with wall distance. Turbulence generation is highest in the direction perpendicular to the flow ($w$-component). In addition, Nienaltowska also explained that aerodynamic sound production occurs when vortex lines are stretched or accelerated relative to the acoustic medium.

Zhu et al. (1993, 1994) obtained from their numerical study using commercial CFD software SCRYU with different A-pillar slant angle at different velocities that the sound intensity level increases with vehicle speed. The numerical study was time dependent conducted at time steps of 10 mili-seconds, which correspond to courant number of 0.4. Furthermore, they found that the A-pillar aerodynamic noise was generated through both vortex generation and breakdown process. Moreover, high-pressure fluctuations seem to be occurring at the front side window and roof side junction with pressure fluctuations at the roof junction much more higher compared to the ones generated behind the A-pillar and occurring higher at higher A-pillar slant angles. Results obtained were similar to the study conducted by Hanaoka (1993). No validations of predicted numerical results were made with empirical data.
Dobrzynski et al. (1994) conducted experiment on the A-pillar region at different yaw angles. He found that huge surface pressure level reduction can be achieved when using smooth A-pillar contour with large radius and flushed side windows compared to using large radius A-pillar with recessed side window.

Hamel et al. (1996) include side mirrors on his A-pillar experiment with various height and vehicle velocities. The addition of side mirror in the experiment increases fluctuating pressure level as much as 20 dB (from 110 dB to 130 dB) at lower frequency downstream close to the side mirror (below 1kHz). No significant increase in sound pressure level close to the A-pillar as a result of side mirror addition. However, only with the presence of side mirror, increasing the A-pillar height increases the peak sound pressure level at low frequency by about 4 dB at both locations downstream of the side mirror and close to the A-pillar.

George et al. (1997) mentioned in his paper that even if the automobile were totally streamlined, external noise would still occur due to the existence of turbulent boundary layer over the vehicle exterior from flow separation.

Uchida et al. (1997) conducted simulation by using CFD (commercial software, SCRYU) to demonstrate the capabilities of using solution adaptive grids in modeling A-pillar flow with side mirrors. Although they managed to show an improved vortex generation from their simulation, they did not justify their findings with any validations against experimental data. The study was more of a parametric study for qualitative observation and focused on showing the effectiveness of the solution adaptive grid techniques for mesh refinements and further evaluation and behaviour of airflow behind the A-pillar region was not discussed.

Bergamini et al. (1997) conducted both experimental and numerical simulation on an A-pillar bluff body at 100 km/h. The study was done to determine the feasibility of numerical simulation in predicting vehicle aerodynamic noise. Numerical simulation was done using unsteady RANS simulation (at a time step of 10 μ-seconds for 0.2 seconds and a courant number of 100) with a one-equation
low-Reynolds Point-Wise Rt turbulence model to predict vehicle noise source. Results from the CFD simulation was then solved using CAA technique (Ffowcs-Williams and Hawkings equation of aerodynamic noise prediction around a solid body). Results from the CAA simulation was compared to experimental results at four different points on the car roof. Power spectral results from CAA simulation showed poor prediction at points close to the edge of the roof (region of separation). The sound pressure level from the numerical results was over predicted (115 dB from measurement compare to 135 dB from predicted). Fair prediction at points on the roof region much downstream to the flow was obtained. Predicted peak sound pressure level matched against values obtained experimentally (at 110 dB) with power spectra pattern fairly captured.

Her et al. (1997) conducted a parametric study using a combination of results from CFD simulation (commercial software, STAR-CD) by using standard k-ε turbulence model and experimental data to get a more accurate result by not using excessive amount of grid generation. Although there was indication that vehicle noise prediction can be carried out using the proposed technique, the current study failed to show any significant improvement in correlation with experimental data.

In 1998, Alam et al. experimented with scale vehicle models exposed at various wind tunnel velocities at different yaw angle and found some Reynolds number dependency at low velocity (40 and 60 km/h). Furthermore, they observed that there is a larger separation on the leeward side with associated velocity drop and increased turbulence intensity.

Strumolo et al. (1998) simulated external aerodynamic noise generated around a simplified wedge box model (to mimic A-pillar region) at 115 km/h by using CFD. Comparison of sound pressure level spectra was made between results obtained from CFD and data obtained experimentally. Results showed that the highest sound pressure level was obtained from vortex flow behind the wedge box (behind A-pillar). Results from CFD simulation slightly over predict results obtained empirically by about 5 dB (125 dB obtained from CFD and 120 dB obtained empirically). In addition, a difference between 30 dB was obtained from
region radiating the lowest aerodynamic noise (95 dB in front of windshield) compared to region of vortex flow.

Uchida et al. (1999) conducted a transient CFD simulation with commercial CFD software, PowerFLOW, using Lattice Boltzmann techniques to predict surface fluctuating pressures at 100 km/h. CFD simulation was first conducted using an initial simplified model representing a vehicle A-pillar. Simulation was conducted at time steps of between 22.5 to 45.0 μ-seconds, with 8.38 to 12.13 million computational grids generated. Measurement points were located behind the A-pillar region. Results of power spectra from the CFD simulation were compared to results obtained experimentally. The peak sound pressure level (below 500 Hz) from the CFD results was slightly over predicted by around 10 dB (120 dB from CFD results and 110 dB from experimental results). However, the overall trend of the power spectra plot was well captured from results using CFD simulation. CFD simulation (using simplified A-pillar case mesh configuration) was then carried out to a full-scale vehicle model. Simulation was conducted at 22.5 μ-seconds time steps with a total amount of 17.36 million computational grids generated. Pressure spectra results comparison against experimental data does not compare as well as the first case. CFD results over predicted peak sound pressure level (at around 250 Hz) by 20 dB (85 dB from experimental results to 105 dB from CFD results). Author concluded this due to the complexity of the model and the lack of mesh resolution. Both cases of CFD simulation showed high sound pressure level at the base of the A-pillar region.

Kumarasamy et al. (1999) conducted experimental and numerical simulation of A-pillar rain gutter to predict aerodynamic noise. CFD simulation was conducted with CFD commercial software PAMFLOW using time dependent Navier-Stokes equation (with Smogorinsky Sub-grid Scale (SGS) algebraic turbulence model) with the CAA simulation carried out using Lighthill-Curle formulation. Simulation was carried out at 80 km/h using 1.7 million computational grids. Furthermore, the simulation was conducted at 6 μseconds time steps of a total time of 15.60 seconds. One million time steps were used. Pressure spectra results of CFD simulation an over prediction of peak sound pressure level by 30 dB (100
dB from experimental and 130 from CFD simulation). However, overall power spectra plot showed that the CFD results under predicted the experimental results by 10 dB.

Alam (2000) extended his experimental testing on various windshield radii to see its effects on A-pillar flow behaviour. It was found that the pressure fluctuations have strong dependence on A-pillar radii. This dependence is highest at negative yaw angle (leeward side), followed by 0° yaw angles and least at positive yaw angle (windward side) respectively. Alam also found that the surface mean and fluctuating pressure coefficients are independent of Reynolds number at higher velocities. In line with previous investigations, Alam found that the maximum hydrodynamic pressure fluctuations occur between the areas of separation and reattachment of the A-pillar vortex. In addition, Alam conducted results on modified A-pillar radius (making it smoother). The highlight of the study was based on results plotted on a graph of maximum $C_p$ RMS values at various yaw angles versus windshield radius. The results showed that as the windshield radius increase, the maximum value of $C_p$ RMS for different yaw angle would decrease.

Zimmer (2003) extended Alam’s testing in that he conducted testing of A-pillar aerodynamic and aeroacoustics (including interior noise measurement) effect on full size vehicle. Zimmer conducted both wind tunnel and on road testing. Results obtained by Zimmer for wind tunnel testing showed difference static and fluctuating pressure coefficient values with Alam due to the presence of transient and crosswind during on road testing. Zimmer found an increase interior vehicle noise with an increase of vehicle speed and when subjected to various yaw angles. Pressure spectra results were also obtained for interior vehicle noise and results were similar to Alam in that the low frequency range contributed to the highest noise level. In addition, Zimmer also modified the vehicle with sound insulation protection (not included A-pillar section). Results showed no reduction in interior vehicle noise further proving that the A-pillar area is the main contributor to aerodynamic noise.
Li et al. (2003) conducted experimental and numerical study on reducing external aerodynamic noise on an actual model of a vehicle A-pillar (external side mirror present), before and after rain gutter modification. Modifications were applied by widening the rain gutter. Numerical simulation was conducted using PAM-FLOW commercial CFD software. By using timed dependent Navier-Stokes simulation (0.2 to 0.5 courant number), two modeling strategy was used, the first with finite difference (structured hexahedron mesh) and the second with finite element methods (unstructured tetrahedral mesh). In excess of 10 millions grids were generated for the simulation (4.0 mm first boundary layer spacing used on the structured hexahedron mesh and 0.44 mm spacing on the unstructured tetrahedron mesh). Only finite element method results were presented in the study. No comparison between finite element and finite difference strategy were made. Results showed a 5 to 10 dB (A) over prediction from the finite element simulation (60 dB (A) from experimental and 65 to 70 dB (A) from finite element method). Peak sound pressure level was measure at frequency region of 250 to 500 Hz for experimental results and 500 to 1.3 kHz for the finite element method. However, there was an improvement in aerodynamic noise made after A-pillar rain gutter modifications on both experimental and finite element method results.
1.10 Conclusions and Evaluation from Previous Research Work

From past literature reviews, it can be concluded that researchers have focused on three main areas in studying A-pillar vortex or vortex separation in general. These areas are:

- The effect of geometry modification of A-pillar in reducing aerodynamic noise.
- The study of physics and mechanics of aerodynamic noise generation in vortex flow.
- Numerical modeling of vortex flow.

In the first area of study, past literature reviews suggest that aerodynamic noise generated by the A-pillar vortex cannot be eliminated fully, due to the ever presence of boundary layer flow, George et al. (1997). The key in A-pillar design is to achieve minimum aerodynamic noise. Studies by Popat (1991) and Alam (2000) have showed that this can be achieved by designing A-pillar with a much more inclined angle (lower than critical angle of 40° to 50°) with a larger windshield radius. However, design, safety concerns and constraint dampers this progress (Alam, 2000). Hence, majority of the current vehicle A-pillar and windshield design adopts an A-pillar inclination angle of around 60° with minimum windshield curvature, (together with a narrow rain gutter on some vehicle models) giving it a quasi-sharp edge, adapting an overall small ellipsoidal radius. This type of design, together with the presence of vehicle add-ons such as the side rear view mirror, results in the generation of complex three-dimensional turbulent flow separations behind the A-pillar. The complex A-pillar vortex generation will result in A-pillar aerodynamic noise caused by large eddies of dipole sound source at low frequency range (below 500 Hz), Haruna et al. (1992). Provided that the vehicle is well sealed, this type of low frequency noise will cause vehicle occupants to feel tired and fatigue especially after a long drive.
In the second area of study, past literatures have also examined the physics and mechanics of the separation vortex and suggested that peak $Cp$ RMS occurs at the core of the vortex, Watanabe et al. (1978), Alam (2000). Study also suggests that the source of effective pressure fluctuations is from the shear layer above the re-circulating flow, Fricke (1968, 1971), Simpson (1987, 1989). However, the mechanism of aerodynamic noise formation was suggested not predominantly from vortex mixing (Laufer, 1974) but also from all facet of the flow i.e. separation, reattachment and vortex breakdown, Hussain (1983, 1986). In addition, past literature also suggests that large-scale turbulent motion is responsible for the generation of aerodynamic noise by changing the mean flow-field to produce low frequency pressure fluctuation at low Mach number, Simpson, (1987, 1989).

In the third area of study, past literature reviews suggests that with the availability of high performance computers and advanced CFD and CAA software’s, trend on A-pillar aerodynamic development studies is slowly shifting from a more experimental approach (using wind tunnel) to a more numerical/computational approach. Computational approach is arguably the way of the future in promising faster turn around simulation time with cheaper running cost. At the same time, computational approach offer superior capability than the experimental approach in terms of post processing of data and graphical representation of flow analysis. Provided computational approach to be reliable and can produce quantifiable results with a high degree of accuracy, the role of wind and aeroacoustics tunnel testing in the future might be reduced for only validation purposes. However, more work should be done on improving CFD and CAA technological advancement for a more efficient and easier way to tackle problems associated with aerodynamics and aeroacoustics.

Past literature reviews on work done in studying vortex flow from computational methods can be divided into two areas. The first area involves steady state simulation in analyzing wall mean properties of vortex flow. The second area involves transient simulation in analyzing the fluctuating properties of the vortex.
flow. Acoustics properties of the vortex flow will then be extracted from the transient simulation results by using relevant CAA techniques.

Literature reviews showed that there has been very little work done in research of vortex flow through computational methods from past 15 years. The author has found that in steady state flow simulation, past researchers rely heavily on one particular turbulence model (standard $k$-$\varepsilon$), which is well known to produce inaccurate results, Wilcox, (2001). Since results from steady state analysis are used as a boundary condition for the transient simulation, a proper comparative study between relevant turbulence models must be needed to carry out in order to ensure a proper solution is obtained. This will in turn provide an accurate solution as a boundary condition for transient simulation, which will provide a good prediction of the aeroacoustics properties.

In the area of transient simulation, work and analysis done from previous literatures involves mainly describing the distribution of surface fluctuating pressure of the unsteady flow, Haruna et al. (1992), Hanaoka et al. (1993). It is believed that there is insufficient information on transient behaviour of A-pillar vortex flow. Analysis has to be carried out in order to explain the behaviour of transient A-pillar vortex flow by describing the corresponding behaviour of its turbulent properties.

It can also be observed from past literature review that in order to predict appropriate surface fluctuating pressures for noise source or aeroacoustics evaluation from the transient simulation, the spatial and temporal properties of the flow must be adequately resolved and therefore grid generation around the area of interest must be sufficiently fine, Uchida et al. (1999), Li et al. (2003). A simulation of this kind will take quite some time to finish, even with the help of parallel computing. Even so, predicted aeroacoustics results from past research study could only provide reasonable correlation with empirical data, Haruna et al. (1992), Bergamini et al. (1997), Uchida et al. (1999), Kumarasamy et al. (1999), Li et al. (2003). The author believes that this proves to be an expensive exercise. Considering that only reasonable correlation can be obtained against empirical
data, a new CAA technique developed by AVL, which evaluates aeroacoustics properties from the steady state CFD RANS results, might produce similar correlation accuracy against empirical data but at a faster turn around time. In addition, CAA results from past research study still contain insufficient information on the aeroacoustics behaviour from A-pillar vortex flow. Although it is widely known from spectral analysis that the source of aerodynamic noise from A-pillar vortex flow lies in the low frequency region, other properties such as distribution of sound pressure level behind the A-pillar region is still unclear.

In addition to steady state and transient CFD simulation, and CAA simulation, another important area is that of grid generation. According to Miller et al. (1999), decision on which grid generation strategy to be used must be determined correctly since it will not only save time but also influence the accuracy of the CFD results. Previous literatures only provide a specific grid generation technique (Haruna et al. (1992), Hanaoka et al. (1993), Uchida et al. (1997, 1999), Makowski et al. (2000), Li et al. (2003)) and failed to provide proper comparative study between various grid techniques and its applicability to A-pillar vortex simulation.
1.11 Research Project Background and Significance, Scope and Proposed Methodology

This PhD project was developed as part of the motivation to extend earlier research work conducted by Alam. He completed his PhD project in 2000 and based his PhD project on earlier research work done by Popat (1991). Popat conducted an experimental investigation on the effect of windshield angle on the A-pillar vortex structure and aerodynamic noise using simplified small-scale models. Alam’s extended Popat’s work in his PhD project by conducting an experimental investigation the effect of A-pillar vortex structure and aerodynamic noise by using simplified small-scale models of varying windshield curvatures and subjecting them at different yaw angles. Both Popat and Alam conducted their research project at three different wind velocities (to simulate vehicle traveling speed). Using experimental methods, visualization of the vortex structure behind the A-pillar region is somewhat limited. The size and structure of the vortex that was developed from various A-pillar windshield radii exposure at various yaw angles remains unknown and is a major motivation behind this project. Using CFD approach, the size and structure of the A-pillar vortex can be observed and quantified. The mechanism of A-pillar vortex generation, the transient and also acoustical behaviour can be investigated and understood.

It is within the interest and scope of this PhD research project to extend the completed PhD research project of Alam. The methodology used for this PhD project will be based on a full CFD approach. The validation of predicted CFD results will be done with experimental data obtained from the completed PhD research project of Alam.

This PhD research project will be conducted using two commercial CFD software provided by Victorian Partnership in Advance Computing (VPAC) and Swinburne University of Technology namely FLUENT and AVL respectively. Computer Aided Drawing (CAD) software will be used to generate simplified
vehicle models of Alam. The CAD models will then be exported to FLUENT and AVL for meshing, solving and post processing.

All five simplified vehicle model used in Alam’s PhD project will be used for this PhD project. For this PhD project, all five simplified vehicle model will be yawed at three different angles and will be exposed to three different wind velocities.

The CFD simulation for this PhD project will be divided between the FLUENT and AVL CFD software. FLUENT and AVL will be used for comparison between grid generation technique and steady state simulation of turbulence model. Furthermore, FLUENT and AVL will also be used for transient simulation. In addition, CAA simulation will be conducted by using AVL.

1.12 Objectives of PhD Project

The objectives of this PhD project are as follows;

- To investigate the most feasible grid generation technique that can be used to get the best possible computational fluid dynamics simulation results for different simplified vehicle model of various windshield radii.

- To investigate the effectiveness of selected turbulence models to best capture A-pillar vortex flow. Results of steady mean flow properties will be compared against available experimental data. Any deficiency obtained from simulation will be further analyzed.

- To establish an appropriate numerical CFD model for better prediction of steady state flow behind the A-pillar region of different simplified vehicle model of various windshield radii.

- To use CFD model from steady state flow to develop CAA model to simulate aerodynamic noise behind the A-pillar region. Results of unsteady mean properties will be compared against available experimental
data. Power spectral density results will be compared against available experimental data.

- To use CFD model to analyse A-pillar vortex flow for various windshield radii and yaw angles.

- To use CAA model to analyse the aeroacoustics behaviour behind the A-pillar region.

### 1.13 Thesis Layout

The layout of this PhD thesis is as follows:

Chapter 1 describes the general background information, the relevant literature review, conclusion and evaluation of previous research work, the motivation and scope and the objectives of this PhD project.

Chapter 2 describes in detail the governing equations and boundary conditions used in this PhD project.

Chapter 3 describes in detail the methodology for this PhD project. In addition, this chapter presents the comparison between grid generation techniques and feasible grid generation technique used for simplified vehicle model with different windshield radii.

Chapter 4 presents results obtained from steady state CFD simulation. This includes comparison of predicted CFD results using selected turbulence models against experimental data, detailed analysis and discussion of results. In addition, appropriate numerical model will be established for simplified vehicle model with different windshield radii.

Chapter 5 describes the CAA preprocessing procedures for this PhD project. This chapter also presents the results obtained from the CAA simulation. Results of
power spectral density and $C_p$ RMS will be compared against available experimental data. In addition, this chapter presents the analysis of transient airflow and aeroacoustics behaviour and distribution behind the A-pillar region.

Chapter 6 presents the major conclusions of this PhD project. In addition, recommendations for further work will also be presented.
Chapter Two

GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

In this chapter, a general overview of computational fluid dynamics (CFD) is presented. This is followed by a general overview of computational aeroacoustics (CAA). Finally, this chapter concludes with the choice of CFD and CAA technique to be used for this research project.

2.1 Turbulence and early works of turbulence modelling

According to Taylor (1935) and Von Karman (1938), turbulence is defined as:

“... an irregular motion which in general makes its appearance in fluids, gaseous or liquid, then they flow past solid surfaces or even when neighbouring streams of the same fluid flow past or over another.”

In 1959, Hinze redefined turbulence based on Taylor and Karman in a more precise manner as:

“Turbulence fluid motion is an irregular condition of flow in which the various quantities show a random variation with time and space coordinates, so that statistically distinct average values can be discerned.”

In addition, Cebeci and Smith in 1974 extended the definition of turbulence as:

“... turbulence has a wide range of scales.”

In general, based on the definitions quoted, it can be said that a turbulent flow exhibit characteristics of irregularity with varying random scales in time and
space that refers to time, velocity and length scale, mixing of large and small eddies (shows diffusivity and dissipation) and is three dimensional in nature.

The earliest account of establishing a numerical description on turbulence can be traced back to Joseph Boussinesq (1877). Boussinesq postulated an eddy-viscosity approximation theory, which leads to the assumption that viscous shear stresses behave in proportional to the mean rate of strain. From molecular gas mixing (Brownian motion) at a molecular level, the instantaneous velocity can be defined as \( u = U + u'' \), where \( U \) is the mean velocity and \( u'' \) is the random fluctuating velocity of molecules. By ensemble averaging the molecular gas transport of momentum, Boussinesq managed to obtain the viscous shear stress as:

\[
t_{ij} = -\rho u''_i u''_j
\]  
(2.1)

Where \( t_{ij} \) is the molecular viscous shear stress tensor, \( u''_i \) and \( u''_j \) are random molecular velocity motion in the \( x \) and \( y \) direction.

Furthermore, based on the Maxwellian distribution for perfect gas, Boussinesq then derived an approximation for molecular viscous shear stress \( (t_{ij}) \) in terms of the product of an eddy viscosity and the mean strain-rate tensor. The molecular viscous shear stress is given as:

\[
t_g = 2\mu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij}, S_{ij} = \frac{1}{2} \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right]
\]  
(2.2)

Where \( \mu_t \) is defined as Boussinesq-Eddy Viscosity for molecular motion, \( S_{ij} \) is the mean strain rate tensor and \( \delta_{ij} \) is the kronecker delta, where it is 0 for \( i \neq j \) and 1 for \( i = j \). The Boussinesq-Eddy Viscosity is defined as:

\[
\mu_t = \frac{1}{2} \rho v_{mfp} l_{mfp}
\]  
(2.3)
Where $v_{th}$ is the thermal velocity and $l_{mfp}$ is the characteristic length of the mean free path of the molecules. It follows that for the Boussinesq-Eddy Viscosity approximation to stand valid, $\mu_t$ must be in accordance to the mean characteristics of the flow.

Reynolds in 1894 proposed a statistical approach in turbulence by time averaging the Navier-Stokes equations. The time averaged Navier-Stokes equations is popularly known as the Reynolds Average Navier-Stokes Equations (RANS). The motivation behind this approach is due to the fact that most cases of turbulence fluid flow involved only the mean (average) properties of the flow. However, by averaging the Navier-Stokes equations, six new terms are introduced. This new term is defined as the Reynolds Stress term and is given by:

$$\tau_{ij} = -\rho u_i' v_j'$$  \hspace{1cm} (2.4)

The Reynolds stress term is analogous to the molecular viscous shear stress approximation from Boussinesq.

### 2.2 Governing Transport Equation and Turbulence Models

The governing mathematical equations used in CFD are equation of continuity (conservation of mass), equation of momentum (Navier-Stokes) and equation of energy (energy conservation). However, for automotive flow problem involving external aerodynamic flow, only the continuity and momentum equations are that of concern.

The continuity equation will be written as a combination of the transient and advection/convection term:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \quad (2.5)
\]

The momentum equation will be written as a combination of the transient, advection/convection, diffusion and source term:

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + S_i \quad (2.6)
\]

In automotive flow problems, practical turbulence problems account more on the mean flow effect. Therefore, on most problems with turbulence flow, a statistical approach is used, by time averaging the momentum and continuity equation to take into account the mean velocity of the flow. In the time averaging approach, the instantaneous velocity of the flow will be divided into the mean and fluctuating components \( (u_i = U_i + u_i) \).

The momentum equation is then transformed to the Navier-Stokes equation (for incompressible fluid) and is written in simplified conserved form as:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial (\rho u_i)}{\partial x_j} = - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + S_i \quad (2.7)
\]

where the diffusion term in its original form is written as:

\[
\frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( 2\mu S_{ij} - \frac{2}{3} \mu \frac{\partial u_i}{\partial x_j} \delta_{ij} \right) \quad (2.8)
\]

The last term in the right represents the effect of volume dilation. After time averaging the mean and fluctuating velocity components, we will obtain the Reynolds Averaging Navier Stokes (RANS) equation:
\[
\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial (\rho \overline{u_i})}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left(2 \mu S_{ij} - \frac{2}{3} \mu \frac{\partial u_i}{\partial x_j} \delta_{ij}\right) + \frac{\partial}{\partial x_j} \left(-\rho \overline{u_i u_j}\right) + S_i \tag{2.9}
\]

Where, \(\overline{u_i u_j}\), is the Reynolds stress term. The corresponding time averaged continuity equation will then be written as:

\[
\frac{\partial \overline{u_j}}{\partial x_j} = 0 \tag{2.10}
\]

For a three-dimensional flow, the system of equations will consist of three RANS equations and a continuity equation, which totals up to four equations. However, with four equations, the system needs to close up for ten unknowns. This consists of one mean pressure, three mean velocity and six Reynolds stresses. Therefore, additional equations are needed to close the system. Closure of the system can be obtained through modelling of the Reynolds stress term in the RANS equation. This is also known as turbulence modelling. The realism of flow simulated by the RANS equations will be dependent on the turbulence model used.

Turbulence models based on RANS equations can be categorised into two groups. The first group uses the ‘eddy-viscosity’ concept by modelling the Reynolds stresses based on the Boussinesq eddy-viscosity approximation. The second group is the non-eddy viscosity group, which attempts to solve for the Reynolds stresses directly. The eddy-viscosity turbulence models can be further classified into several groups:

- Algebraic (zero-equation) models
- One-equation models
- Two-equation models
2.3 Algebraic (zero equation) Turbulence Models

The most simplistic turbulence model is the algebraic turbulence model. It is also known as the zero-equation turbulence model. Prandtl (1925) was responsible for the origin of the algebraic turbulence model. Prandtl’s work was inspired by the earlier work of Boussinesq (1877). Turbulence mixing length scale will be prescribed from values obtained empirically, limiting the application of the algebraic turbulence model. Therefore, different values of the turbulence mixing length are needed for different type of flow application. The accuracy of the predicted flow will depend on the prescribed turbulence empirical mixing length scale. Because of this, the algebraic turbulence models perform at its best when the local turbulence flow is behaving in an isotropic manner.

The first algebraic model that was developed by Prandtl (1925) was the mixing length model. The mixing length model is a simple model and was first developed for boundary layer type flow. It was described in equation 2.11 as:

\[
\tau = \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \mu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \rho \tau_{mix} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)
\]  

(2.11)

To close the equation, the value for \( \tau_{mix} \) must be prescribed experimentally and it varies with different type of flow application.

Important modifications on the mixing length model were also made by Theodore Von Karman (1930), Van Driest (1956), Clauser (1956) and Corrsin et al. (1954) and Klebanoff (1954) to improve on Prandtl’s mixing length equation. Smith-Cebeci (1967) and Baldwin-Lomax (1978) then developed their own algebraic turbulence model by using and combining ideas proposed on improving the algebraic model made by researchers prior to them. The end product was a two layer algebraic model developed for boundary layer type flow. According to Wilcox (2002), although both models are equally superior, the Baldwin-Lomax model is more popular among researchers due to its robustness.
2.4 One Equation Turbulence Models

Reflecting back on the algebraic turbulence model, as simplistic as it was, its deficiencies can cause non-physical solution at certain flow application. It is important to take into account the turbulent scales in order to develop a more complete turbulence model. Proper turbulence historical profile upstream of the flow will be needed to properly calculate the flow characteristics of the turbulence profile downstream to the flow. This resulted in the development of the one equation turbulence model. In this model, the turbulent velocity scales are modeled via a transport equation for mean turbulence kinetic energy in order to close the system.

Prandtl realized the deficiencies of the mixing length model. To further improve on his previous work on mixing length model, in 1945 he proposed the inclusion of the time-averaged turbulent kinetic energy as a turbulent property in order to determine a better turbulent eddy viscosity. By taking the trace of the Reynolds stress tensor, the turbulent kinetic energy (per unit mass) is defined as:

$$\tau_{ij} = -u_i u_j = -2k$$  \hspace{1cm} (2.12)

Which will lead to the definition of the isotropic turbulent kinetic energy as:

$$k = \frac{1}{2} \mu u_i u_i = \frac{1}{2} (u_i^2 + v_i^2 + w_i^2)$$  \hspace{1cm} (2.13)

From this Prandtl came to a conclusion that the turbulent eddy kinematic viscosity is defined as:

$$\nu_{t} = k^{1/2} l$$  \hspace{1cm} (2.14)
The turbulent kinetic energy in equation 2.14 will be solved through a transport equation. The transport equation for the turbulent kinetic energy is:

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \nu \left( \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ v \frac{\partial k}{\partial x_j} - \frac{1}{2} u_i u_j - \frac{1}{\rho} p u_j \right] \right) \tag{2.15}
\]

In order to close for the transport equation for the turbulence kinetic energy, the Reynolds stress in the production term, the dissipation term and the turbulence and pressure diffusion term needs to be prescribed. For the Reynolds stress, the one equation still utilises Boussinesq eddy viscosity approximation.

Since Boussinesq eddy viscosity approximation is still being used to determine the Reynolds stress, the one model equation still holds the same deficiency of the algebraic turbulent model in associating mean velocity gradient in shaping the turbulent property of the flow.

The Prandtl (1945) one-equation model was also attributed to Kolmogorov, Emmons (1954) and Glushko (1965). Kolmogorov was attributed due to his involvement in their effort to develop the model. Emmons on the other hand coincidently proposed the same one equation model in 1954. Apart from the Prandtl one equation model, a few notably one equation turbulence model was developed in the 1960s by Bradshaw, Ferriss and Atwell (1967), Nee and Kovasznay (1968) and Wolfstein (1967 and 1969).

Recent efforts to improve the one equation model have been made by Baldwin et al. (1990) and Spalart et al. (1992). These one-equation turbulence models are dubbed complete since the need to prescribed empirical values for the length scale is no longer needed.

The Spalart-Allmaras turbulence model only showed a slight advantage against the algebraic turbulence model. Prediction of boundary layer flow near the wall is within close similarity with the algebraic turbulence model. Separated flow
showed a slight better prediction by the Spalart-Allmaras turbulence model over the algebraic turbulence model. However, the Spalart-Allmaras turbulence model still suffers from setbacks. It has problems in predicting jet flows, separated and decaying turbulence flow (Wilcox, 2001).

In summary, it can be said that although it is a good move to include turbulent kinetic energy as a representation of the turbulent velocity scale, the one equation model is only slightly better than the algebraic turbulence model and suffer from the lack of proper modelling of the turbulence length scale. Both zero and one-equation models are good in modelling isotropic equilibrium flows in region of low Reynolds number the production of turbulence and dissipation are in balance. However, in non-equilibrium flows, properly modelling needs to be done to accommodate the rapid changes in turbulence length scale.

2.5 Two Equation Turbulence Models

In the two-equation turbulence model, in addition to the turbulence velocity scale (mean turbulence kinetic energy), the turbulence length scale is also modelled via a transport equation. The modelled turbulence length scale will then be added together with the mean turbulence kinetic energy in the turbulence eddy viscosity for a better prediction of the Reynolds stresses using the Boussinesq approximation. Several version of the modelled turbulence length scale exists and it will be later explained in this section. Launder et al. (1972a) described the generalised form of the modelled length scale as:

\[ z = k^n l^m \]  \hspace{1cm} (2.16)

Where \( m \) and \( n \) are constants.

Kolmogorov proposed the first two-equation model in 1942. Kolmogorov proposed the \( z \) equation to be defined as the specific dissipation rate, \( \omega \). Ever since Kolmogorov postulated his two-equation model, several researchers have
come up with their version of the two-equation model. A summary of the two-equation models is provided in the table below.

Table 2.1: Versions of the two-equation models

<table>
<thead>
<tr>
<th>Researcher(s), Year</th>
<th>( z )</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotta (1951), Spalding (1967)</td>
<td>( l )</td>
<td>( l )</td>
</tr>
<tr>
<td>Rotta (1968, 1971), Rodi-Spalding (1970), Ng-Spalding (1972)</td>
<td>( kl )</td>
<td>( kl )</td>
</tr>
<tr>
<td>Zeierman-Wolfshtein (1986), Speziale-Abid-Anderson (1990)</td>
<td>( k^{-1/2}/l )</td>
<td>( \tau )</td>
</tr>
<tr>
<td>Spalding (1969), Robinson-Harris-Hassan (1995)</td>
<td>( k/l^2 )</td>
<td>( \omega^2 )</td>
</tr>
</tbody>
</table>

Out of all the versions listed in Table 2.1, the \( k-\omega \) model and the \( k-\varepsilon \) model have been popular and widely used. The focus in this section will be on these two equations models. Details on the other version of the two-equation models are available from Launder et al. (1972a) and Wilcox (2002).

According to Wilcox (2002), Kolmogorov came up with his proposed transport equation for \( \omega \) through dimensional analysis and reasoning. Kolmogorov transport equation for \( \omega \) was defined as:

\[
\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = -\beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \sigma \frac{\partial \omega}{\partial x_j} \right]
\]  

(2.17)

Which represents the convective term, the dissipation term and also the diffusion term. Wilcox (2002) described Kolmogorov equation should have included the
term for production. Although Kolmogorov first postulated that $\omega$ represented dissipation of the smallest eddies and therefore should not contain a production term, it was later found out large eddies are actually responsible in determining the time scale of turbulence, hence the rate of dissipation. Furthermore, Kolmogorov equation did not include a molecular diffusion term making it only valid for high Reynolds number flow and cannot be integrated through the viscous sub-layer region of the flow.

The Kolmogorov $k-\omega$ equation was then modified and further developed by researchers (Refer Table 2.1) over the years and the most popular version was the one by Wilcox (2002), which was defined as:

$$
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (v + \sigma^* V_T) \frac{\partial k}{\partial x_j} \right] 
$$

(2.18)

Where the closure coefficients are obtained empirically, Wilcox (2002). Both the closure coefficients and auxiliary relations are further defined as:
The Wilcox $k-\omega$ turbulence model is an empirical turbulence model and was first developed as a high Reynolds number turbulence model. It was then later that a low Reynolds number version of the Wilcox $k-\omega$ was developed, Wilcox (2002), Fluent (2003).

As with the $k-\omega$ turbulence model, the $k-\varepsilon$ turbulence model exists in many versions. Jones et al. developed the current standard version of the $k-\varepsilon$ turbulence model in 1972, Launder et al. (1972a). The model of Jones and Launder was developed for high Reynolds number application but accounted for low Reynolds number effects as well. Launder et al. (1974) later improved the
The standard $k-\varepsilon$ turbulence model is defined as:

\[

\nu_T = C_\mu \frac{k^2}{\varepsilon}
\]

\[

\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T / \sigma_k) \frac{\partial k}{\partial x_j} \right]
\]

\[

\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = C_{\epsilon1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{\epsilon2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T / \sigma_\varepsilon) \frac{\partial \varepsilon}{\partial x_j} \right]
\]

(2.21)

$C_{\epsilon1} = 1.44, C_{\epsilon2} = 1.92, C_\mu = 0.09, \sigma_\varepsilon = 1.0, \sigma_k = 1.3$

$\omega = \varepsilon / (C_\mu k), l = C_\mu k^{1/2} / \varepsilon$

The $k-\varepsilon$ turbulence model predicts the kinematic eddy viscosity value through the modelling of the turbulence velocity and length scale. This is achieved by solving additional transport equation of mean turbulent kinetic energy (velocity scale) and dissipation rate (length scale).

Over the years the standard $k-\varepsilon$ turbulence model have proved to be both economical and robust. However, it has its deficiencies, such as the inability to predict highly strained flow, swirling flow, rotating and separating flow, Naser (2003). However, due to the deficiencies of the standard $k-\varepsilon$ turbulence model, improvements have been made to overcome such setbacks, which lead to the development of the RNG $k-\varepsilon$ model and the Realizable $k-\varepsilon$ model.
Yakhot et al. (1986) developed the RNG $k - \varepsilon$ turbulence model in 1986 based on a statistical formulation called the Renormalization Group Theory. The main objective of the development of this turbulence model was to modify the kinematic eddy viscosity and the dissipation transport equation so that they are able to adapt to highly strained flows, the effect of swirl flows and to account for low Reynolds number effects, Fluent (2003).

The RNG $k - \varepsilon$ turbulence model is defined as:

$$\mu_r = \rho C_\mu \frac{k^2}{\varepsilon}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right)$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = C_{\mu} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{2\varepsilon} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right)$$

(2.22)

Where the closure coefficients and auxiliary functions are given as:

$$C_{2\varepsilon} = C_{2\varepsilon} + \frac{C_\mu \lambda^3 (1 - \lambda / \lambda_o)}{1 + \beta \lambda^3}$$

$$C_{\mu} = 1.42, C_{2\varepsilon} = 1.68, C_\mu = 0.085, \sigma_i = \sigma_e = 0.72$$

$$\beta = 0.012, \lambda_o = 4.38$$

(2.23)

For highly strained mean flow, the RNG $k - \varepsilon$ turbulence model modified the closure coefficient in the dissipation term of the dissipation transport equation. Instead of taking $C_{2\varepsilon}$ as a constant, it has been modified to include mean strain effect, which can be seen from equation 2.23. In highly strained mean flow, $C_{2\varepsilon}'$ will be smaller than $C_{2\varepsilon}$, thus reducing the effect of dissipation in the dissipation term. This will result in a reduction in the mean turbulent kinetic energy effect and therefore reducing the turbulent eddy viscosity value, $\mu_r$, Fluent (2003). This will reduce the over diffusive behaviour, experienced in the standard $k - \varepsilon$ turbulence model.
As with the standard $k - \varepsilon$ turbulence model, the RNG $k - \varepsilon$ turbulence model take into account for low Reynolds number effects. The diffusion term on both transport equation was modified for this matter. For low Reynolds number flow, the integration of equation 2.24 and the inverse effective Prandtl numbers, $\alpha_k$ and $\alpha_\varepsilon$ is used to account for the varying Reynolds number. The calculation from equation 2.24 will then be incorporated into the diffusion term for both transport equation, and will also be used to determine the value for the turbulence eddy viscosity, $\mu_r$. For high Reynolds number flow, the default $\mu_r$ from equation 2.22 will be used.

\[
d\left(\frac{\rho k}{\sqrt{\varepsilon \mu}}\right) = 1.72 \frac{\hat{\nabla} \cdot \hat{\nabla}}{\sqrt{\nabla^2 - 1 + C_v}} d\hat{\nabla}, \hat{\nabla} = \frac{\mu_{\text{eff}}}{\mu}, C_v = 100
\]

\[
\frac{\alpha - 1.3929}{\alpha_o - 1.3929} \left(\frac{\alpha + 2.3929}{\alpha_o + 2.3929}\right)^{0.6321} = \frac{\mu}{\mu_{\text{eff}}}, \alpha_o = 1.0
\]

\[
\frac{\mu}{\mu_{\text{eff}}} = 1, \alpha_k = \alpha_\varepsilon \approx 1.393
\]

For swirling effects, the RNG $k - \varepsilon$ turbulence model incorporates the function from equation 2.25 into the turbulence eddy viscosity, $\mu_r$.

\[
\mu_r = \mu_o f\left(\alpha_s, \Omega, \frac{k}{\varepsilon}\right)
\]

Where $\mu_o$ can be taken from either high or low Reynolds number flow formulation for turbulence eddy viscosity, $\mu_r$.

However, although the RNG $k - \varepsilon$ turbulence model is an improvement from the standard $k - \varepsilon$ turbulence model, it still has its limitations. This is due to the transport equation for the dissipation term, $\varepsilon$. Because of the fact that the
transport equation for the dissipation term is derived based on physical reasoning and dimensional analysis alone, certain mathematical limitation will exist and needs to be satisfied in order to be consistent with the prediction of Reynolds stresses, hence the turbulence flow, Fluent (2003). The RNG and standard \( k-\varepsilon \) turbulence model does not satisfy these mathematical limitations, making them not ‘Realizable’. Due to this fact, Shih et al. developed the Realizable \( k-\varepsilon \) turbulence model in 1995 to address these mathematical limitations and improve on it. From their research, they have developed a new transport equation for the dissipation term from the exact equation based on the mean-square vorticity fluctuation transport equation, Shih et al. (1995). In addition, they also modified the formulation for the turbulence eddy viscosity term, \( \mu_T \), by improvising on the closure constant \( C_\mu \), varying it in order to adjust to different flow applications.

The Realizable \( k-\varepsilon \) turbulence model is defined as:

\[
\nu_T = C_\mu k^2 / \varepsilon \\
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_j \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial k}{\partial x_j} \right] \\
\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = C_\mu S \varepsilon - C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial \varepsilon}{\partial x_j} \right]
\]

(2.26)

Where the closure coefficients and the formulation for \( C_\mu \) are defined as:

\[
C_1 = \max \left[ \frac{0.43, \eta}{\eta + 5} \right], \eta = S \frac{k}{\varepsilon}, C_2 = 1.9, \sigma_k = 1.0, \sigma_\varepsilon = 1.2 \\
C_\mu = \frac{1}{A_s + A_i} \frac{kU^*}{\varepsilon}, A_s = 4.04, A_i = \sqrt{6 \cos \phi}, \\
\phi = \frac{1}{3} \cos^{-1} \left( \sqrt{6} W \right), W = \frac{S_y S_{\beta} S_{\nu}}{S}, \bar{S} = \sqrt{S_y S_{\nu}}, S_y = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \\
U^* = \sqrt{S_y S_{\nu} + \Omega_y \Omega_{\nu}} \\
\Omega_y = \Omega_{\nu} - \varepsilon_{ijk} \omega_k
\]

(2.27)
From equation 2.26 it can be seen that the transport equation of the dissipation term is different from the standard and RNG $k-\varepsilon$ turbulence model. The main difference lies in the production and dissipation term. According to Fluent (2003), the dissipation term in the dissipation transport equation was modified so that the denominator will not be singular, eliminating the chance of the dissipation term of becoming too big. This will provide a reasonable value for dissipation in the $k$ transport equation and therefore overcoming the problem of non-decaying eddy viscosity, Wilcox (2002). Furthermore, according to Shih et al. (1995), the modification of the production term will encourage a better transfer of spectral energy. Both modification of the dissipation transport equation term promotes a better description of the turbulent vortex stretching, which is required to better predict free shear flows, especially the spreading rates of jets, Wilcox (2002). In addition, the closure coefficient $C_\mu$ was not made constant. It will correspond to different flow application. As was described in Shih et al. (1995), boundary layer flow will yield a $C_\mu$ value of 0.09, as per measured value. For homogeneous shear flow, the $C_\mu$ will yield a value of 0.06, which is close to the measure value of 0.05. However, it can be seen from Shih et al. that since the Realizable $k-\varepsilon$ turbulence model is primarily designed for high Reynolds number flow, providing good prediction of free shear flows. However, its performance close to the boundary layer especially under the influence of pressure gradients is not very good. Only free shear flows yields better prediction than the standard $k-\varepsilon$ turbulence model.
2.6 Deficiencies of the Two Equation Turbulence Model

Deficiency of the two-equation model can be traced to three main reasons, which is the inability to predict flow in the presence of pressure gradient close to the wall and due to the inability to take into account the effect of rapid change in mean strain rate due to the sharp streamline curvature of the geometry.

2.6.1 Pressure Gradient Effects

The two-equation model, especially the $k$-$\varepsilon$ still fail to accurately predict flow in the boundary layer region, close to the wall even after treating it. Perturbation of the defect layer then showed that the reason for this was due to large turbulence scale developed close to the boundary layer especially in the presence of adverse pressure gradient effect. Analysis by Wilcox (2002) showed that by adding a cross diffusion term to the $k$-$\varepsilon$ turbulence model, the increase of the turbulence scale close to the wall could be suppressed. Furthermore, analysis of wake Coles et al. (1969) wake strength parameter in the defect layer against a function of pressure gradient showed that as adverse pressure gradient effect increases close to the wall, the wake strength parameter was being under predicted as a result of an increase in skin friction coefficient close the wall.

According to Rodi et al. (1986), two-equation turbulence model perform badly in region close to the wall due to the fact that the dissipation length scale rises sharply near the wall. However, Bradshaw (1969) have shown through experimental data that the dissipation length scale is independent in the region close to the wall. Coles et al. (1969) also supported this and described that the turbulent boundary layer close to the wall exists in different scales and physical processes. Rodi et al. (1986) also suggest that under adverse pressure gradient, the rate of kinetic energy production is more than the rate of dissipation itself, resulting in an increase in turbulence dissipation length scale. This will lead to an increase and over prediction of turbulent eddy viscosity and wall shear stress.
However, although the $k$-$\varepsilon$ versions of the two-equation model have proved inaccurate in predicted a wide range of flow due to its deficiencies, the $k$-$\omega$ turbulence model developed by Wilcox (2002) have proved to be rather successful. Part of the reason is the presence of the cross diffusion term in the $k$-$\omega$ as a way to suppress the increase of the turbulence dissipation length scale close to the wall. However, the addition of cross diffusion term can cause instability to the calculation of the flow, since it augments the influence of convection in the flow. Therefore, treatment is needed to curb this instability, Wilcox (2002). The $k$-$\omega$ turbulence model prediction of boundary layer flow only differ by 3.5% from measurement and have shown to predict separation rather successfully as well. However, a minor setback of the $k$-$\omega$ turbulence model is that the sub-layer region has to be integrated which means grid generation near the wall must be sufficiently fine.

2.6.2 Effect of Rapid change of Mean Strain Rate and Streamline Curvature

According to Wilcox (2002), the main flaw of the eddy viscosity model, which assumes Boussinesq approximation is the fact that it assumes the Reynolds stress are proportional to the mean strain rate at all part of the flow, a postulation from Stokes based on laminar flows. The constant of proportionality was defined as the eddy turbulent viscosity. However, turbulent flow is affected by the change of geometry, anisotropy and flow history. With the presence of curvature in the flow, the individual Reynolds stress values will be different and hence, again the Boussinesq assumptions will become flawed, Wilcox (2002).

Bradshaw (1973) described that eddy-viscosity models offers inaccurate prediction of flows that undergo rapid changes in rates of strain. The main reason for this is because the adjustment of Reynolds stresses to such changes is unrelated to turbulence mean flow and time scale effect.
Therefore, the effect of curvature can be treated either by making correction to the two-equation models. Alternatively, the Reynolds Stress Transport model can be used. Naser (1990) described from Bradshaw (1973) that since the effect of curvature causes large changes in the Reynolds stresses and triple velocity correlations, the Boussinesq assumptions must be corrected. Bradshaw (1973) proposed that the extra effect of strain rate would modify the Boussinesq assumption to include a factor of an order of magnitude higher. Therefore, the Reynolds stresses from the Boussinesq assumption will be modeled as:

\[
\overline{u'v'} = \nu_r \left( \frac{\partial \bar{u}}{\partial y} + \alpha \frac{\partial \bar{v}}{\partial x} \right)
\]  

(2.28)

Where \(\alpha\) is of the order of 10.

Modifying certain aspect in the two-equation turbulence model can also treat the effect of flow curvature. Leschnizer et al. (1981) made modifications in the \(C_\mu\) to take into account the effect of curvature and obtained good prediction in parallel jet flow. Furthermore, Bradshaw (1969) proposed a dimensionless number that takes into account the curvature effect. Sharma (1975) and Launder et al. (1977) modified the dissipation equation in the \(k-\varepsilon\) to take into account the curvature effect. With that respect, they modified the \(C_{\varepsilon_2}\). Rodi (1979) made corrections in the \(C_{\varepsilon_1}\) constant. Finally, Wilcox et al. (1977) proposed a correction to the \(k-\omega\) turbulence model within the \(k\) equation term and the logarithmic law equation.

### 2.7 Reynolds Stress Turbulence Models

Deficiencies of the eddy viscosity turbulence model inspired early turbulence models developer such as Chou (1945) and Rotta (1951) to propose a turbulence model where the Reynolds stresses, \(\overline{u_iu_j}\) are not modeled through the Boussinesq assumption and the Reynolds stresses, are solved directly through a set of transport equations. This turbulence model is known as the Reynolds Stress
Turbulence Model (RSM) or second order/moment closure. According to Wilcox (2001), such early contributors to the development of the RSM include Donaldson et al. (1968), Daly et al. (1970), Launder et al. (1975), Gibson et al. (1978), Lumley (1978), Speziale (1985, 1987a, 1991) and Reynolds (1987). Throughout the years, Launder et al. (1975) RSM have became the one of the most popular and have been set as a baseline for RSM.

The direct use of Reynolds stresses to for turbulence modeling, especially in a three-dimensional flow, generates six extra transport equations, which represents the individual components of the Reynolds stresses. In addition, a transport equation is also used to solve for the turbulence length scale and this is normally employed through the use of the dissipation transport equation, $\varepsilon$.

The transport equation for Reynolds stress from Wilcox (2002), is given as:

$$\frac{\partial \tau_{ij}}{\partial t} + U_k \frac{\partial \tau_{ij}}{\partial x_k} = -\tau_{ij} + 2\nu \frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \left[ \nu \frac{\partial u_i}{\partial x_j} + 
\right] \quad (2.29)$$

Equation 2.29 consists of convective term on the left hand side of the equation. The first two terms on the right hand side of the equation is the production term, followed by the dissipation term, two pressure strain redistribution term and the diffusion term. For a three-dimensional flow, the six extra Reynolds stress transport equations generate extra 22 unknowns. Ten unknowns generated are in the form of triple velocity correlation that forms the turbulent diffusion term. A further six unknowns are generated from the dissipation term and six more unknowns from the pressure strain distribution term, Wilcox (2002).

According to AVL (2003), the extra unknowns generated are of higher order correlations and the main challenge in Reynolds stress modelling is to model these unknowns. The terms that need to be modelled consists of the dissipation term, the turbulent diffusion term and the pressure strain redistribution term. According to Wilcox (2002), the resulting Reynolds stress equation will be:
\[
\frac{\partial \tau_{ij}}{\partial t} + U_k \frac{\partial \tau_{ij}}{\partial x_k} = P_{ij} + \varepsilon_{ij} - \Pi_{ij} + \frac{\partial}{\partial x_k} \left[ \nu \frac{\partial \tau_{ij}}{\partial x_k} + C_{ijk} \right]
\]  
(2.30)

Where the production \( P_{ij}, \varepsilon_{ij}, \Pi_{ij} \) and \( C_{ijk} \) are given as:

\[
P_{ij} = -\tau_{ik} \frac{\partial U_j}{\partial x_k} - \tau_{jk} \frac{\partial U_i}{\partial x_k} \\
\varepsilon_{ij} = 2\nu \frac{\partial u_i}{\partial x_k} \frac{\partial u_j}{\partial x_k} \\
\Pi_{ij} = \frac{p}{\rho} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \\
\rho C_{ijk} = \rho u_i u_j u_k + \rho u_i \delta_{jk} + \rho u_j \delta_{ik}
\]
(2.31)

Which consists of the production term, the dissipation term, the pressure strain redistribution term and the turbulent diffusion (triple velocity correlation) term.

According to Wilcox (2002), the Reynolds stress component in the production term, \( P_{ij} \), is solved through the Reynolds stress transport equation.

The dissipation term, \( \varepsilon_{ij} \), is modelled based on high Reynolds number flow conditions where the effect of \( \varepsilon_{ij} \) can be assumed to be isotropic (Hanjalic et al., 1976) and is defined as:

\[
\varepsilon_{ij} = \frac{2}{3} \varepsilon \delta_{ij}
\]
(2.32)

Where, \( \delta_{ij} \) is the boundary layer thickness, which accounts for the effects of near wall anisotropy. The transport equation used to solve for \( \varepsilon \) is similar to the dissipation transport equation from the standard \( k-\varepsilon \) turbulence model. Hanjalic et al. (1972) and Launder et al. (1975) defined their transport equation for \( \varepsilon \) as:
The turbulent diffusion term, $\rho C_{ijk}$, which consists of higher order velocity fluctuation correlation components was firstly modelled by Daly et al. (1970) using a simple gradient-diffusion hypothesis, which defines $C_{ijk}$ to be:

$$C_{ijk} = C_s' \frac{k}{\epsilon} \tau_{km} \frac{\partial \tau_{ij}}{\partial x_m}$$

(2.34)

Hanjalic et al. (1972) and Launder et al. (1975) proposed another version of the modelled $C_{ijk}$ term, which was defined as:

$$C_{ijk} = C_s \frac{k}{\epsilon} \left( \tau_{im} \frac{\partial \tau_{jk}}{\partial x_m} + \tau_{jm} \frac{\partial \tau_{ik}}{\partial x_m} + \tau_{km} \frac{\partial \tau_{ij}}{\partial x_m} \right)$$

(2.35)

The constants $C_s'$ and $C_s$ are given approximately as 0.25 and 0.11 respectively. However, according to Lien et al. (1994), the equation derived in 2.34 and 2.35 might have destabilising effect especially in three-dimensional flows. Therefore, an isotropic version of $C_{ijk}$ is provided and defined as:

$$C_{ijk} \equiv C_s \frac{k v^2}{\epsilon}$$

(2.36)

Where $C_s$ was defined as 0.22 and $v^2 \equiv 0.361k$.

Shir (1973) have also come up with a simple gradient diffusion concept, in which $C_{ijk}$ is defined as:
The last term that requires modelling for the Reynolds stress turbulence model is
the pressure strain term. According to Wilcox (2002), the pressure strain term is
the term that received the most attention. The reason for this is because since the
pressure strain term has the same order as the production term, it plays an
important role in the majority of engineering flows of interest. In addition, the
pressure strain term requires some degree of creativity to model in order to
achieve approximate closure.

For homogeneous turbulence, the pressure strain term is divided into two parts, a
slow fluctuating pressure and a rapid fluctuating pressure.

\[
\Pi_{ij} = \phi_{ij,1} + \phi_{ij,2}
\]  

(2.38)

Where the first term on the right hand side of equation 2.38 represents the slow
pressure strain rate term and the second term represents the rapid pressure strain
term.

Rotta (1951) postulated that the slow pressure term could be modelled linearly as:

\[
\phi_{ij,1} = C_1 \frac{\epsilon}{k} \left( -\tau_{ij} + \frac{2}{3} k \delta_{ij} \right)
\]  

(2.39)

Where \( C_1 \) is a closure coefficient obtained from the empirical measurement of
Uberoi (1956) to lie between 1.4 < \( C_1 \) < 1.8.

Launder et al. (1975) proposed a model to solve for the rapid pressure strain
redistribution term based on the analysis of Rota (1951), which is strictly valid
only for homogeneous turbulence. Gibson et al. (1978) then proposed a simplified
version of the Launder et al. (1975) model, which defines the rapid pressure strain redistribution term as:

\[
\phi_{ij,2} = -C_2 \left( P_{ij} - \frac{2}{3} \delta_{ij} P \right)
\]

\[
P_{ij} \equiv -\tau_{ik} \frac{\partial U_j}{\partial x_k} - \tau_{jk} \frac{\partial U_i}{\partial x_k}
\]

\[
P \equiv -\tau_{ik} \frac{\partial U_j}{\partial x_k}
\]

Speziale et al. (1991) have developed a simple, non-linear pressure strain redistribution model for incompressible flow, which is often called as the SSG quadratic pressure strain model. The SSG model has become a popular choice among pressure strain redistribution model since it does not require a correction for the near wall reflection effect (will be explained in the following section). The SSG model is defined as:

\[
\Pi_{ij} = -\left( C_1 e + C_1^* \tau_{mn} \frac{\partial U_m}{\partial x_n} \right) b_{ij} + C_2 e \left( b_{ik} b_{ij} - \frac{1}{3} b_{mn} b_{mn} \delta_{ij} \right)
\]

\[
+ \left( C_3 - C_3^* b_{ij} b_{ij} \right) k S_{ij} + C_4 k \left( b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij} \right)
\]

\[
+ C_5 k \left( b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik} \right)
\]

Where, \( b_{ij} \) is defined as:

\[
b_{ij} = \frac{\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij}}{\tau_{mn}}
\]

And the constants are given as:

\[C_1 = 3.4, C_1^* = 1.8, C_2 = 4.2, C_3 = 0.8, C_3^* = 1.3, C_4 = 1.25, C_5 = 0.4\]
The main advantage of the RSM is that it takes into account the anisotropic nature of turbulence in predicting flows by incorporating non-local and history effect of turbulence, Ahmed (1998), Wilcox (2002). Due to this, RSM provide better prediction of flow over streamline curvature, swirling flows, rotation, and flow with rapid changes in the mean strain rate, FLUENT (2003). According to Wilcox (2002), unlike eddy viscosity turbulence models, which uses treatment and compensation through the manipulation of mathematical formulation to predict complex flows, the RSM provides better prediction of complex flow in a more natural way by direct modeling of the Reynolds stresses.

Furthermore, according to AVL (2003), RSM modeling of each individual stress component enables accurate prediction of anisotropic turbulence, which often plays a crucial role as a major source of energy in complex flows, such as in stagnation regions, secondary motion and in vortex flows. RSM also provide accurate prediction of the unequal wall-normal stress component, which is important in reproducing wall shear stress and mass transfer close to the wall.

However, despite the advantages that have been showed by the RSM in predicting complex flows, it does not offer any superiority over eddy viscosity models in predicting free stream flows and boundary layer flow and it also share the shortcomings experienced by the eddy viscosity model especially in poor prediction of three dimensional and separated flow. This deficiency can be traced back on the modelling of higher correlation terms such the pressure strain redistribution term and also on the modelling turbulence length scale through the dissipation ($\varepsilon$) term, Wilcox (2002), FLUENT (2003). According to Wilcox (2002) the usage of the dissipation term in predicting the turbulence length scale will inadvertently transfer the deficiency of the two equation eddy viscosity models to the Reynolds stress turbulence model. Wilcox (2002) has also shown from the DNS simulation of Parneix et al. (1998) that deficiency of the RSM can also be traced back through the modelling of the pressure strain rate correlation. The DNS simulation showed inaccurate results was obtained despite accurate modelling of dissipation, $\varepsilon$. The inability for Reynolds stress turbulence model to provide better prediction for three-dimensional flows can be traced back on the
application of two-dimensionality in boundary layer flows. According to AVL (2003), current turbulence models have been developed based on the knowledge of two-dimensional flows. In three-dimensional boundary layer, the eddy viscosity is not isotropic. Latest work of Schwarz et al. (1994) has studied the performance of some Reynolds stress turbulence model in three-dimensional boundary layer flow. Based on their findings, they have concluded that the main cause of error in the performance of RSM models in a three dimensional boundary layer is due to the inadequacy of the modelling of dissipation.

A new and attractive alternative to the RSM is the use of hybrid turbulence model that incorporates attributes from the eddy viscosity two-equation models and the RSM. The hybrid turbulence model offer the advantages of the RSM turbulence model in terms of its accuracy and its applicability over curved flows and at the same time offer the robustness and convergence speed of the eddy viscosity two equation model. For this instance, Basara et al. (2003) proposed a hybrid turbulence model (HTM) that combines both the advantages and applicability of the RSM and $k-\varepsilon$ turbulence models. In the HTM model proposed by Basara et al. (2003), the solution for the Reynolds stress is obtained form the Boussinesq assumption of equating stress to the mean strain of the flow (equation 2.10). The solution for the turbulence eddy viscosity is obtained similarly from the k-epsilon model (equation 2.73). The calculation of $k$ and $\varepsilon$ to solve for the turbulent eddy viscosity is obtained similarly from the RSM turbulence model (Refer the AVL use of RSM turbulence model). The $k$ equation obtained from the summation of normal Reynolds stresses. The dissipation term is solved from the transport equation for dissipation. The main difference in the HTM turbulence model is the $C_\mu$ constant used in defining the turbulent eddy viscosity. $C_\mu$ is defined by:

$$C_\mu = \left[-u_i u_j \frac{\partial U_i}{\partial x_j} \right] \left( \frac{k^2}{\varepsilon} S^2 \right), S = \sqrt{2S_{ij}S_{ij}}$$

(2.43)

Which is the ratio between the RSM production term and the $k-\varepsilon$ production term. The rationale from the variable value of $C_\mu$ is due to the fact that according
to Basara et al. (2003) that $C_\mu$ varies not only from flow to flow but also varies at different points in the same flow. $C_\mu$ is defined as:

$$C_\mu = \left(\frac{-\langle u_i u_j / k \rangle}{\mu}\right)^2$$

was originally determined to be around 0.09, but was later discovered to vary somewhere in between 0.06 to 0.12 depending on the type of flow. Validation of HTM against various flow applications has showed results that lie in between RSM and $k-\varepsilon$ turbulence model. Deficiencies of the HTM turbulence model is due to the inability of the HTM to predict anisotropic Reynolds stresses, which is due to the use of eddy-viscosity coupling of the Reynolds stresses with the mean velocity.

### 2.8 Direct Numerical and Large Eddy Simulation

The ideal method to obtain accurate results for turbulence flow is to directly solve for the Reynolds stresses and in turn the non-linear Navier-Stokes and continuity equations. This can be achieved by using either the method of Direct Numerical Simulation (DNS) or by using Large Eddy Simulation (LES). DNS and LES are also known as the unsteady viscous methods, Ahmed (1998).

In DNS, unsteady Navier-Stokes equations are solved directly without any modelling of the turbulence. The fluctuation velocity and viscous force components within the body surface and in the computational domain is obtained together with components of Reynolds stresses. On the other hand, the LES approach directly solves the large eddies motions using the unsteady Navier-Stokes equations and models the small eddies through the use of a sub-grid scale filters (linear or Gaussian). The motivation behind this lie on the fact that small eddies has a more universal character. They are more isotopic and dissipative in nature, which makes its behaviour independent to the flow. Large eddies are
highly anisotropic and unsteady in nature, which makes it dependent to the flow, Ahmed (1998).

However, according to Ahmed (1998), the LES approach is a more preferred option for technical flows because it requires significantly less computer capacity than DNS, and yet at the same time promises to be more accurate and robust than the conventional RANS approach.

At the moment, DNS is used purely for research purposes. According to Wilcox (2002), current DNS application is mostly limited to Low Reynolds number in two dimensional and simple three-dimensional flows (incompressible and compressible) with homogeneous turbulence. The extremely high demand that DNS put on computational capacity leads back to the extremely high grid generation needed to resolve all turbulent scale (even the smallest Kolmogorov turbulence scale) in both spatial and temporal dimension.

According to Ahmed (1998), DNS requires grid points that increase with $\frac{9}{4}$ power of Reynolds number. It has been estimated that to capture the effect of the smallest turbulence motion and solves the flow around a structure the size of a vehicle, grid points around $10^{18}$ would be needed. By the mid 1990s, the largest number of grid points that could be handled computationally was only around $10^6$, which makes it so impractical in using DNS vehicle aerodynamics in the immediate future, Barnard (1996).

Due to this, the approach of using LES is better suited than using DNS. According to Wilcox (2002) and FLUENT (2003), LES grid size can be at least one order of magnitude smaller than DNS. Furthermore, a large time step can be used with flow achievable at a higher Reynolds number. Available LES models are from the Smagorinsky (1963) simple gradient-diffusion model, the Lilly (1966) one equation model and the second order closure of Deardorff (1973).
2.9 CFD Near Wall Treatment and Boundary Conditions

Turbulence models discussed in the preceding section was mainly developed for flows of high Reynolds number values. For laminar boundary layer flow near to the wall, molecular viscous effects that dominate the turbulent eddy viscosity must be accounted for, Ahmed (1998). According to Wilcox (2002), one of the main reason for high Reynolds number turbulence model (most two-equation, HTM and RSM models) inability to predict accurate values for flow near the wall especially in the presence of adverse pressure gradients was traced down to the inaccurate reproduction of the law of the wall constant, $C$. Analysis showed that in the example of a standard $k-\varepsilon$ turbulence model, predicted $C$ value of –2.2 as oppose to the measured value of 5.0. In another analysis conducted by Rodi et al. (1986), the deficiency of high Reynolds number turbulence model can be traced back to the deficiency in the model of the dissipation rate, $\varepsilon$ to predict an accurate length scale value near to the wall. According to Rodi et al. (1986), for most high Reynolds number turbulence model, the production of turbulence kinetic energy, $k$ and length scale near to the wall is too steep, resulting from a lower production in the rate of dissipation. This will result in a high value of Reynolds stresses near the wall region. In order to overcome this problem, proper treatment is needed to compensate for this deficiency, for a more reliable near wall prediction.

Two different approaches are often used for near wall treatment. They are:

- Wall Function approach
- Low Reynolds Number Model approach
2.10 Wall Function Approach

The first algebraic model that was developed by Prandtl (1925) was the mixing length model. The mixing length model is a simple model and was first developed for boundary layer type flow. It was described in equation 2.45 as:

\[ \tau_{ij} = \mu_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \mu_i = \rho l_{mix}^2 \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \]  

(2.45)

To close the equation, the value for \( l_{mix} \) must be prescribed experimentally and it varies with different type of flow application. Theodore Von Karman in 1930 produced a similarity hypothesis based on Prandtl mixing length model. Through similarity rule, Von Karman expresses \( l_{mix} \) as:

\[ l_{mix} = \kappa \left| \frac{dU}{dy} - \frac{d^2U}{dy^2} \right| \]  

(2.46)

Where \( l_{mix} \) is prescribed in terms of space coordinates and \( \kappa \) is the universal turbulence constant or Karman constant, which can be determined experimentally. However, according to Launder et al. (1972a), Von Karman similarity hypothesis is only valid at area very close to the wall. With this, Prandtl proposed an assumption that the mixing length is proportional to the distance from the wall \( (y) \) when \( y/\delta < 0.20 \). \( \delta \) is defined as the boundary layer thickness. Therefore \( l_{mix} \) is then defined as:

\[ l_{mix} = \kappa y \]  

(2.47)

Where \( \kappa \) in equation 2.47 was later determined through experimental as approximately as 0.41.

Von Karman similarity hypothesis can then be simplified to:
\[ U = \frac{u_*}{\kappa} \ln y + C \]  
(2.48)

The constant \( C \) can be further evaluated to be:

\[ \frac{U}{u_*} = \frac{1}{\kappa} \ln \frac{u_* y}{\nu} - \frac{1}{\kappa} \ln \beta \]  
(2.49)

Where, \( \frac{U}{u_*} \), is the ratio of the mean flow to the friction velocity and is defined as \( u^* \). \( \frac{u_* y}{\nu} \), is the dimensionless distance from the wall and is defined as \( y^* \).

Equation 2.49 is famously known as the law of the wall or the universal velocity-distribution law. Therefore, equation 2.49 can be simplified further to be:

\[ u^* = \frac{1}{\kappa} \ln y^* + 5.0 \]  
(2.50)

The first and the most common approach in treating near wall effect are by using the wall function approach of Launder et al. (1974). The wall function approach uses a semi-empirical model, which assumes the behaviour of fluid flow near the wall to follow the law of the wall profile, which will enforce the proper value for \( C \) in the wall region. Another advantage for using the wall function approach is a shortened computational time since the steep flow gradients occurring in the sub-layer region does not need to be resolved. This allow for coarse grid to be generated close to the wall. For the first cell next to the wall, the wall function approach automatically assumes a law of the wall profile, which is valid in the region of \( y^* < 30 \) to 500, Roberson et al. (1997). The wall function approach of Launder et al. (1974) is defined as:

\[ \frac{U_p}{\tau_w / \rho} \frac{C_{\mu}^{1/4} k_p^{1/2}}{\nu} = \frac{1}{\kappa} \ln \left( E_{yp} \frac{C_{\mu}^{1/4} k_p^{1/2}}{\nu} \right) \]  
(2.51)
Where, $E$ is given as the wall roughness, which is equal to 9.0 for a smooth wall, Launder et al. (1974). In order to evaluate for the kinetic turbulence energy near the wall, $k_{\mu}$, the average energy dissipation rate is deduced from the following equation:

$$
\int_0^{y_+} \varepsilon \, dy = C_\mu \left( \frac{k_p^{3/2}}{\kappa} \ln \left( \frac{C_{\mu}^{1/2} k_p^{1/2}}{\nu} \right) \right)
$$

In addition, for $y^+ < 11.225$, a laminar stress-strain relationship is used and is defined as:

$$
\frac{U_p}{\tau_w / \rho} C_{\mu}^{1/4} k_p^{1/2} = y_p \frac{C_{\mu}^{1/4} k_p^{1/2}}{\nu}
$$

However, the standard wall function has its disadvantages. According to FLUENT (2003), due to the fact that the standard wall function was developed from two-dimensional flow (Ahmed, 1998) and rely on ideal conditions (constant shear and local turbulent equilibrium), its will yield inaccurate results when conditions deviate too much from ideal. The wall function approach does not perform very well in conditions of severe pressure gradients close to the wall, highly three-dimensional flow close to the wall and where turbulent effect is strongly non-equilibrium. Throughout the years, improved versions of the standard wall function have been developed to include non-equilibrium and pressure gradient effects, which offer better prediction for complex flows that includes separation and reattachment, FLUENT (2003).

Kim et al. (1995) developed such wall function approach, often known as the non-equilibrium wall function. The wall function model can withstand non-equilibrium flow and pressure gradient effects and was developed based on a two-layer model concept proposed by Chieng et al. (1980) and Launder (1988).
adjacent to the wall are assumed and divided into a viscous sub layer and a fully
turbulent layer. The non-equilibrium wall function is defined as:

\[
\frac{\tilde{U}}{\tau_w / \rho} C_{\mu}^{1/4} k^{1/2} = \frac{1}{\kappa} \ln \left( E \frac{C_{\mu}^{1/4} k^{1/2}}{\nu} \right)
\]  

(2.54)

Where, \( \tilde{U} \), takes into account the pressure gradient effect and is defined as:

\[
\tilde{U} = U - \frac{1}{2} \frac{d\rho p}{dx} \left[ \frac{y}{\rho \kappa C_{\mu}^{1/4} \sqrt{k}} \ln \left( \frac{y}{y'_v} \right) + \frac{y - y'_v}{\rho \kappa C_{\mu}^{1/4} \sqrt{k}} \right] 
\]  

(2.55)

And the sub-layer thickness, \( y_v \) is defined as:

\[
y_v = \frac{V}{C_{\mu}^{1/4} k_p} y'_v 
\]  

(2.56)

Furthermore, the turbulent kinetic energy, \( k \), is defined as:

\[
k = \begin{cases} 
\left( \frac{y}{y'_v} \right)^2 k_p, & y < y'_v \\
0, & y > y'_v 
\end{cases}
\]  

(2.57)

According to Makowski et al. (2000), the non-equilibrium wall function is
recommended for vehicle aerodynamic computations because it compensate for
higher than ideal values of \( y^+ \) near to the wall and also account for the effects of
the variation in wall adjacent grid when computing for the turbulence kinetic
energy, near to the wall. Furthermore, the non-equilibrium wall function provides
a more realistic prediction of the behaviour of turbulent boundary layers
(including separated flow) without increasing significant computational resources.
In addition to the non-equilibrium wall function model of Kim et al. (1995), a similar wall function model was developed by Basara et al. (2003) and was validated against RSM and HTM turbulence models.

2.11 Low Reynolds Number Model approach

The second method in treating near wall effect is by using a low Reynolds number model approach. The main difference between the wall function approach and the low Reynolds number model approach lies in the grid generation near to the wall. In the low Reynolds number model approach, the grid generation in the normal component to the wall must be small enough to resolve flow down until the sub-layer region of the flow. High Reynolds number turbulence model that employs this approach undergoes modification to include damping functions particularly in the dissipation equation ($\varepsilon$) and in the turbulent eddy viscosity term, Ahmed (1998). Popular low Reynolds number model that has been developed includes the low Reynolds number model of Jones et al. (1972), Launder et al. (1974), Lam et al. (1981) and Chien (1982).

According to Wilcox (2002), for a two-equation turbulence model, the modification of the transport equation of dissipation near the wall and turbulent eddy viscosity term are defined as:

$$
\nu_r = C_f \frac{k^2}{\varepsilon}
$$

$$
U \frac{\partial k}{\partial x} + V \frac{\partial k}{\partial y} = \nu_r \left( \frac{\partial U}{\partial y} \right)^2 - \nu + \frac{\partial}{\partial y} \left[ \left( \nu + \frac{\nu_r}{\sigma_k} \right) \frac{\partial k}{\partial y} \right]
$$

(2.58)

Equation 2.58 was developed by Lam et al. (1981), where, the $\varepsilon$ term is defined as $\varepsilon = \varepsilon_o + \tilde{\varepsilon}$, in which the $\tilde{\varepsilon}$ term is solved through the modified dissipation transport equation. The damping functions generated for the low Reynolds
number to take place are given as $f_1, f_2, f_\mu, \epsilon_\mu$ and $E$. In addition, as described previously, the application of the damping function and the low Reynolds number models are dependent on the turbulence Reynolds number and $y^+$ values near the wall.

In certain cases, a one-equation model is preferred in the near wall region to couple with either the high Reynolds number two-equation or RSM turbulence model. This was due to the fact that the empirical nature of the one-equation model often provides good prediction for near wall flow. Popular one-equation model used for near wall model approach include Wolfstein (1969) and Norris et al. (1975). This kind of approach is also known as the two-layer model approach, FLUENT (2003), Naser (1990, 2003).

When the low Reynolds number model is used, the switch from the high Reynolds number turbulence model to the low Reynolds number wall model is normally done when the turbulent Reynolds number (equation 2.59) is less then 200.

$$\text{Re}_T \equiv \frac{\rho \sqrt{k}}{\mu} = \frac{k^2}{\nu \epsilon} = \frac{k^{1/2} l}{\nu}$$  \hspace{1cm} (2.59)

To resolve the flow all the way until the viscous sub-layer, the adjacent grid requirement next to the wall must be $y^+ < 5$ and at least 10 grids generated in the sub-layer region of Reynolds turbulent number lesser than 200, FLUENT (2003).

For the one-equation of Wolfstein (1967, 1969), the turbulent kinetic energy, $k$, equation is and the turbulent eddy viscosity are defined as:

$$\mu_t = C_\mu \rho k^{1/2} l_\mu$$

$$\begin{align*}
\mu \frac{\partial^2 k}{\partial y^2} &+ \mu_t \frac{\partial^2 k}{\partial y^2} + \mu_t \frac{\partial^2 u}{\partial y^2} - C_D \rho k^{3/2} \frac{l_D}{l_\mu} = 0
\end{align*}$$ \hspace{1cm} (2.60)

Where the turbulent and dissipation length scale, $l_\mu$ and $l_D$ are defined as:
In most cases, the length scale in equation 2.60 has been modified to suit the ones given by Chen et al. (1988), which is defined as:

\[
\begin{align*}
I_\mu &= \gamma \left(1 - e^{-Re_{\gamma}/\mu} \right) \\
I_D &= \gamma \left(1 - e^{-Re_{\gamma}/\mu} \right)
\end{align*}
\] (2.61)

For the RSM turbulence model, the low Reynolds number flow effect is incorporated in the modelled dissipation term and also in the pressure strain redistribution term, Wilcox (2002). Manceau et al. (2002) have introduced an elliptical blending model to take account of the near wall low Reynolds number effect. The model was based on the elliptical relaxation model of Durbin (1993) and is defined as:

\[
\phi_j = (1 - k\alpha) \phi_j^w + k\alpha \phi_j^h
\] (2.63)

Where, \(\phi_j^h\) was taken as the pressure strain redistribution model of Speziale et al. (1991) from equation 2.120 and \(\phi_j^w\) is the wall reflection effect defined as:

\[
\phi_j^w = -5\frac{E}{k} \left( \tau_{ik} n_j n_k + \tau_{jk} n_i n_k - \frac{1}{2} \tau_{km} n_i n_m (n_j - \delta_{ij}) \right)
\] (2.64)

However, results have shown that although low Reynolds number model for near wall treatment have managed to reproduce an appropriate C value for the logarithmic law equation, it still does not manage to predict good results for flow close to the wall, Wilcox (2002). It was shown in Wilcox (2002) that in most cases under adverse and favorable pressure gradient application, the low Reynolds
number model still over predicts the values for skin friction coefficients. According to Wilcox (2002), the main reason why the low Reynolds number models failed to predict proper flow behaviour close to the wall is due to the inability of the two-equation model to properly predict the flow in the defect layer, which translate to poor prediction of the flow in the sub-layer.

2.12 Boundary Conditions

In the inlet, outlet and on the wall surface within the computational domain, the boundary conditions for turbulent properties such as the turbulent kinetic energy, \( k \) and the dissipation rate, \( \varepsilon \) must be specified. According to Naser (2003), the inlet boundary condition is defined as:

\[
k = 0.01U^2
\]

\[
\varepsilon = \frac{k^{3/2}}{0.3l}
\]

(2.65)

The outlet boundary condition applied is:

\[
\frac{\partial k}{\partial n} = 0, \frac{\partial \varepsilon}{\partial n} = 0
\]

(2.66)

Where \( n \) is normal to the outlet.

For boundary conditions in RSM, an input for the Reynolds stresses and dissipation rate is required at the inlet of the computational domain, FLUENT (2003). The values can be entered should it be known or derived from the turbulence intensity and characteristic length. According to Naser (2003), at the inlet the boundary conditions are given as:

\[
\overline{u_i u_i} = k, \overline{u_i u_j} = \overline{u_i u_k} = 0.5k, \overline{u_i u_j} = 0
\]

(2.67)
Where, FLUENT (2003), specify the boundary condition value at the inlet as:

\[
0.5\bar{u}_i = 0.5\bar{u}_j = 0.5\bar{u}_k = k \\
\sum u_mu_m = \frac{2}{3}k, u_j = 0
\]  

(2.68)

For the outlet, the boundary conditions are given as:

\[
\frac{\partial (u_i' u_j')}{\partial n} = 0
\]

(2.69)

According to FLUENT (2003), on the wall surface, the no slip boundary conditions for \( k \) is defined as:

\[
\frac{\partial k}{\partial n} = 0
\]

(2.70)

Where \( n \) is the local coordinate normal to the wall. The boundary conditions for the production of \( k \) and \( \varepsilon \) is defined as:

\[
P_k = \tau_w \frac{\tau_w}{\kappa \rho C_{\mu}' \kappa^{1/2} k_p^{1/2} y_p} \\
\varepsilon = \frac{C_{\mu}' k_p^{3/2}}{\kappa y_p}
\]

(2.71)

At the wall, FLUENT (2003) obtained near wall valued of Reynolds stresses and dissipation rate from the wall function. By this, FLUENT assumes turbulence to be in equilibrium. The Reynolds stresses are then computed from:

\[
\frac{u_i'^2}{k} = 1.098, \frac{u_j'^2}{k} = 0.247, \frac{u_k'^2}{k} = 0.655, -\frac{u_i'u_j}{k} = 0.255
\]

(2.72)
Where $\tau$, $\eta$ and $\lambda$ are the tangential, normal and bi-normal coordinate near the wall. Alternatively, the Reynolds stresses near the wall can be specified using the wall shear stress and not $k$. The boundary condition will then be:

$$\frac{u'^2}{u''^2} = 5.1, \frac{u''^2}{u''^2} = 1.0, \frac{u'^2}{u''^2} = 2.3, \frac{u''u''}{u''^2} = 1.0 \quad (2.73)$$

2.13 Computational Aeroacoustics

In retrospect, aerodynamic noise prediction can be obtained from DNS simulation. However, this is not possible due to the impractical demand of DNS on computational resources and the damping nature of most CFD numerical schemes, Lyrintzis (1994), AVL (2003). As an alternative, aerodynamic noise prediction for CAA can be obtained from CFD data of steady state or transient in nature. This normally involves a two-step procedure.

According to Kumarasamy et al. (1999), the first step involves transient CFD calculation of the flow field. Although the generation of aerodynamic noise is a compressible flow phenomenon, incompressibility flow assumption can be used to obtain a solution for subsonic flow with Mach number less than 0.3. The second step involves extracting the acoustical sources term before predicting using relevant CAA numerical technique to predict the propagation of aerodynamic noise.

According to Lyrintzis (1994), Kumarasamy et al. (1999) the CAA numerical technique can be categorized into four:

- Lighthill Acoustic Analogy Method
- Kirchoff Method
- Perturbation Method
- Linearized Euler Equation Method
2.14 Lighthill Acoustic Analogy Method

Majority of study on CAA incorporate the Lighthill Acoustic Analogy to evaluate for aerodynamic noise propagation. The Lighthill Acoustic Analogy was made famous by the late Professor Sir James Lighthill where in 1951 and 1954 introduced papers proposing a theory on aerodynamic noise generation in free stream flow, Goldstein (1976). In a review made by Firoz (2000), Lighthill postulated that all non-linearities in a fluctuating hydrodynamic flow of a large volume of fluid act as sources of sound. The source of aerodynamic noise is given by the Lighthill stress tensor, which is derived thorough the manipulation of the continuity and momentum equation to form an in homogeneous wave equation. This set of formulation is famously known as the Lighthill’s acoustic analogy. In addition, through dimensional analysis, Lighthill’ acoustic analogy have shown to provide a quadrupole type sound source distribution, AVL (2003).

According to Goldstein (1976), the Lighthill’s acoustic analogy equation is defined as:

$$\rho' = \frac{1}{4\pi a_c^2} \frac{\partial^2}{\partial x_i \partial x_j} \int \frac{T_{ij} \left( y, t - \frac{|x-y|}{a_o} \right)}{|x-y|} dy$$  \hspace{1cm} (2.74)

with $T_{ij}$ is the Lighthill’s turbulence stress tensor and is defined as:

$$T_{ij} = \rho_o y_i y_j + \delta_{ij} \left[ (p - p_o) - c_o^2 (\rho - \rho_o) \right] - e_{ij}$$  \hspace{1cm} (2.75)

According to Goldstein (1976), in order to solve for the Lighthill’s acoustic analogy equation, the sound source term, which is the Lighthill’s stress tensor ($T_{ij}$) must be known. However, to obtain a solution for $T_{ij}$ is virtually impossible for most flows of interest since it is equivalent to solving the complete non-linear
equations governing the flow. Therefore, at low Mach number flow, the Lighthill’s stress tensor can be approximated as:

\[ T_{ij} \equiv \rho_v v_i v_j \]  

(2.76)

This is valid for region within the turbulent flow and approximately zero outside the region of turbulent flow.

However, according to Firoz (2000), the Lighthill’s acoustic analogy cannot be applied directly to vehicle noise evaluation. The reason for this is twofold. Firstly, the Lighthill’s acoustic analogy was postulated for a free stream turbulent flow and secondly, the quadrupole type sound source distribution of Lighthill’s acoustic analogy has low efficiency, especially for flow at low mach number. Aerodynamic noise distribution for vehicle occurs in the presence of solid boundary, which represents the vehicle surface. Because of this, efforts were made by Curle (1955) and Ffowcs-Williams and Hawkins (1969) to extend Lighthill’s work to include modeling of aerodynamic noise propagation around a solid body. From equation 2.174, which is the general solution to Lighthill’s acoustic analogy, Curle (1955) derived and extended the Lighthill’s acoustic analogy to include the effect of stationary solid boundaries. The equation is defined as:

\[
\rho' = \frac{1}{4\pi a_o^2} \frac{\partial^2}{\partial x_i \partial x_j} \int_T T_{ij} \left( y, t, \frac{r}{a_o} \right) dy + \frac{1}{4\pi a_o^2} \frac{\partial}{\partial x_i} \int_S P_i \left( y, t, \frac{r}{a_o} \right) dS(y), 
\]

(2.77)

\[ P_i = -l^j p_j \]

In addition, the Curle’s formulation of the Lighthill’s acoustic analogy have shown dipole type sound source distribution, which is much more efficient in comparison to the quadrupole type sound source distribution from the original Lighthill’s acoustic analogy formulation.
Ffowcs-Williams and Hawkins (1969) have also extended Curle’s-Lighthill’s formulation to include flow for moving solid boundaries, for example in flow involving helicopter rotors. According to Kumarasamy et al. (1999), the Ffowcs-Williams and Hawkins formulation is appropriate for high-speed flow applications. For low Mach number flows, the simpler Curle’s-Lighthill’s formulation is recommended for use.

Other notable contributors to the extension of Lighthill’s acoustic analogy for various aerodynamic noise applications include Powell (1964), Howe (1975, 1981, 1993) and Jenvey (1989). Powell’s (1964) contribution involved work on the generation of aerodynamic noise from unsteady vortex flow. On the other hand, Howe (1975, 1981, 1993) was involved in the study of aerodynamic noise generation from turbulent separated flows, especially trailing edge flow. Finally, Jenvey (1989) contributed in the development areas of mean sound power formulation from time-stationary motion.

### 2.15 Kirchoff Method

According to Kumarasamy et al. (1999), the Kirchoff method separates the computational domain into an inner and outer region. The inner is a non-linear region close to the sources and is solved using a CFD solver. The outer is a linear region and is solved using an integral equation for sound pressure level base on the sources distributed on the surface edge of inner region (defined as Kirchoff surface), which can be either stationary or non-stationary. More information on Kirchoff method is available in from Lyrintzis (1994).

### 2.16 Perturbation Method

According to Bergamini et al. (1997), the perturbation method solves the incompressible and acoustic parts of the flow, using two different grids, thus reducing overall computational time. In predicting aerodynamic noise propagation, the difference between the acoustic and the incompressible quantities
are taken as the source terms and solved using an Euler type of equation. More
information on the perturbation method is available from Hardin (1995).

2.17 Linearized Euler Equation Method

The CAA prediction technique using Linearised Euler equation method originally
uses nonlinear/unsteady CFD calculation to predict near-field acoustic source
term (Lighthill’s stress tensor, $T_{ij}$) and uses the linearised Euler equation to
predict the far-field aerodynamic noise propagation, Lyrintzis (1994). However,
according to AVL (2003), in using non-linear CFD approach to determine $T_{ij}$, a
high computational effort is required, in terms of both memory and storage
capacity. Furthermore, according to Lele (1997), formulation will breakdown
whenever significant refractive effects are present in the flow. In addition, a
tailored Green’s function needs to be developed for different geometry to take
account of diffraction effect from interaction between the flow and solid
boundaries.

To overcome the requirement of using non-linear CFD approach to determine $T_{ij}$,
an alternative approach was developed by Bechera (1996), where he used models
based on the space-time correlation function of the turbulent velocity fluctuations.
In his model, Bechera (1996) uses eddy length scale, decay rate and turbulent
kinetic energy data obtained from a steady RANS computation to determine for
$T_{ij}$. Lighthill acoustic analogy is then used to develop a relationship between the
near field and far-field noise correlation, AVL (2003).

Longatte then extended the approach of Bechera (1996) by using a statistical
model formulation instead of using the Lighthill’ acoustic analogy to forge a
relation between the near-field and far-field aerodynamic noise. The approach by
Bechera (1996) is still used to determine a time accurate formulation using the
space-time correlation model with data for acoustic source terms estimation
obtained from steady CFD RANS results. The aerodynamic noise propagation
from the source to the receiver is then modeled using a linearized Euler equation. In addition, refraction and diffraction effects are handled automatically by the linearised Euler equation, AVL (2003).

AVL SWIFT CAA (2003) uses the CAA prediction technique with linearised Euler equation developed by Bechera (1996) and Longatte (1998. This technique employed by AVL SWIFT CAA is a hybrid approach where two separate grids and also calculations are needed for the CFD and CAA prediction. Furthermore, according to AVL (2003), unstructured grid is used for the CAA prediction, which allows for handling arbitrarily complex geometries. In addition, the CAA prediction approach employed by AVL SWIFT is aimed at predicting the broadband noise and cannot be used to predict tonal or periodic noise.

The steps involve in the calculation of CAA in SWIFT is broken down into three, which is:

- Time –averaged CFD simulation with RANS solver
- Acoustic Source Generation
- Acoustic loads-radiated noise with unstructured Linearized Euler Solver

In the first step, the CFD simulation is carried out to determine both the mean flow and turbulence statistics. The mean flow is to be used in step two and three. The turbulence statistics are used to determine the acoustics sources in step two. In the second step, the acoustic sources are estimated from the turbulent fluctuations. In the third step, the propagation of the aerodynamic noise is calculated through the use of the linearised Euler equations through the use of Discontinuous Galerkin Method, AVL (2003).

Once the steady RANS CFD simulation have been conducted, the SWFT CAA determined the acoustic sources from the governing equations of mass, momentum and energy. These equations of mass, momentum and energy are defined as follows:
Mass
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = S_m
\]  
(2.78)

Momentum
\[
\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\rho \vec{U} \vec{U}) + \nabla p = \vec{S}_f
\]  
(2.79)

Energy
\[
\frac{\partial p}{\partial t} + \vec{U} \cdot \nabla p = (\gamma - 1) S_e + \frac{(\gamma - 1)}{2} |\vec{U}|^2 S_M
\]
\[-(\gamma - 1) \vec{U} \cdot \vec{S}_f - \gamma p \left( \nabla \cdot \vec{U} \right) = Q_e - \gamma p \left( \nabla \cdot \vec{U} \right)
\]  
(2.80)

Similar to the generation of the Lighthill’s equations, the mass equation was multiplied by \( \vec{U} \) and subtracted from the momentum equation to obtain:

\[
\frac{\partial \vec{U}}{\partial t} + (\vec{U} \cdot \nabla) \vec{U} = \frac{1}{\rho} \left( \vec{S}_f - \nabla p - \vec{U} S_M \right) = \frac{1}{\rho} \left( \vec{Q}_f - \nabla p \right)
\]  
(2.81)

According to AVL (2003), the mass, momentum and energy equations generated five unknowns, which was density, \( \rho \), the three components of velocity, \( \vec{U} \) and pressure, \( p \). These unknowns are solved using the Linearised Euler Equations (LEE).

The first step in using LEE to solve for density, velocity and pressure was to first divide them into free stream and acoustics components. For velocity, a turbulent component was also included.

Density
\[
\rho = \rho_o + \rho_a
\]  
(2.82)

Pressure
\[
p = p_o + p_a
\]  
(2.83)

Velocity
\[
\vec{U} = \vec{U}_o + \vec{u}_a + \vec{u}_t
\]  
(2.84)
Therefore, the separation of density, pressure and velocity into its free stream, acoustics and turbulent components, transforms the mass, momentum and energy equations.

**Mass**

\[
\frac{\partial (\rho_o + \rho_a)}{\partial t} + \nabla \cdot (\rho_o +\rho_a)(\bar{U}_o + \bar{u}_a + \bar{u}_t) = S_{Mo} + S_{M}'
\]  
\(2.85\)

**Momentum**

\[
\frac{\partial (\bar{U}_o + \bar{u}_a + \bar{u}_t)}{\partial t} + \left( (\bar{U}_o + \bar{u}_a + \bar{u}_t) \cdot \nabla \right) (\bar{U}_o + \bar{u}_a + \bar{u}_t) = \frac{1}{(\rho_o + \rho_a)} \left( \dot{Q} + \dot{Q}_l - \nabla (p_o + p_a) \right)
\]  
\(2.86\)

**Energy**

\[
\frac{\partial (p_o + p_a)}{\partial t} + (\bar{U}_o + \bar{u}_a + \bar{u}_t) \cdot \nabla (p_o + p_a) = \dot{Q}_{so} + \dot{Q}_e - \gamma (p_o + p_a) \left( \nabla \cdot (\bar{U}_o + \bar{u}_a + \bar{u}_t) \right)
\]  
\(2.87\)

In which, the mass, momentum and energy equations are divided into free stream and perturbation (acoustics and turbulent) components. The perturbation components are further defined as:

**Mass**

\[
\frac{\partial \rho_a}{\partial t} + \nabla \cdot (\rho_o \bar{u}_a + \rho_o \bar{u}_t + \rho_a \bar{U}_o + \rho_a \bar{u}_a + \rho_a \bar{u}_t) = S_{M}'
\]  
\(2.88\)

**Momentum**

\[
\frac{\partial \bar{u}_a}{\partial t} + \frac{\partial \bar{u}_a}{\partial t} + (\bar{U}_o \cdot \nabla) \bar{u}_a + (\bar{U}_o \cdot \nabla) \bar{u}_a + (\bar{u}_a \cdot \nabla) \bar{U}_o + (\bar{u}_a \cdot \nabla) \bar{u}_a + (\bar{u}_a \cdot \nabla) \bar{u}_a
\]  
\(2.89\)
Furthermore, according to AVL (2003), the second order and cross terms in the perturbation equations are neglected to further simplify the set of equations.

From the simplified equations of mass, momentum and energy, the left hand side of these equations is solved from the LEE of acoustic density, velocity and pressure respectively. The right hand side of these equations contains sets of source terms. The aerodynamic source generation for mass, momentum and energy are defined as:
The aerodynamic noise source generation are required as part of the input in the acoustic propagation module. It can be seen that in order to solve for the aerodynamic source generation, the turbulent velocity fluctuation term needs to be determined. This is done through reconstruction of the turbulent quantities from the solved steady RANS CFD simulation.

According to AVL (2003), the turbulent velocity can be obtained through the inverse Fourier transform process and can be defined as:

\[ u(x) = 2\sum_{n=1}^{N} \tilde{u}_n \cos(\kappa_n x + \psi_n) \sigma_n \]  

(2.97)

Where, \( \psi_n \), is a random phase, \( \kappa_n \), is the wave-vector and \( \sigma_n \), is the direction of the turbulent velocity vector, which is perpendicular to \( \kappa_n \). \( \tilde{u}_n \) is the mode amplitude and is defined as:

\[ \tilde{u}_n = \sqrt{E(\kappa) d\kappa} \]  

(2.98)

\( E(\kappa) \) is the distribution of energy over the wave numbers and is further defined as:
\[ E(\kappa) = 1.453 \frac{2k}{\kappa^2} \left( \frac{\kappa}{\kappa_e} \right)^4 \left( 1 + \frac{\kappa}{\kappa_e} \right)^{\frac{17}{6}} e^{\frac{2\varepsilon}{\kappa^2}} \]  

(2.99)

Where, \( k \) is the turbulent kinetic energy obtained from the CFD RANS results. \( \kappa_e \) is the peak wave number over the selected wave-number spectrum and \( \kappa_v \) is the Kolmogorov wave number. Both \( \kappa_e \) and \( \kappa_v \) are defined as:

\[ \kappa_e = \frac{0.747}{\Lambda} \]  

(2.100)

and

\[ \kappa_v = \sqrt[4]{\frac{\varepsilon}{\nu^3}} \]  

(2.101)

Where, the dissipation rate, \( \varepsilon \) is determined from the RANS CFD results and the turbulent integral length scale, \( \Lambda \) is defined as:

\[ \Lambda = \Lambda_{fac} \left( \frac{2/3k}{\varepsilon} \right)^{3/2} \]  

(2.102)

The time dependent domain for turbulent velocity is determined from similar relationship as equation 2.180 using inverse Fourier transforms with a time dependent phase and is defined as:

\[ u(x,t) = 2\sum_{n=1}^{N} \tilde{u}_n \cos(\kappa_n x + \psi_n + \omega_n t) \sigma_n = u(x,t) \]

\[ = 2\sum_{n=1}^{N} \tilde{u}_n \cos(\kappa_n (x - Ut + \psi_n)) \sigma_n \]  

(2.103)
Where, the radial frequency, $\omega_n$, is defined as:

$$\omega_n = -\kappa_n U$$  \hspace{1cm} (2.104)

In addition, the coupling between spatial and time domain modifies the energy distribution relationship and is defined as:

$$E(\kappa_n, \omega_n) = E(\kappa_n) \phi_m(\omega_n - \omega_e) = E(\kappa_n) \frac{1}{\omega_n \sqrt{2\pi}} e^{-0.5((\omega_n - \omega_e) / \omega_n)^2}$$  \hspace{1cm} (2.105)

Once the aerodynamic noise source generation has been determined, the final acoustic propagation module is then obtained by using LEE in three-dimensional conversation form.

$$L(U) = \frac{\partial}{\partial t} U + \frac{\partial}{\partial x_j} F_j(U) = S$$  \hspace{1cm} (2.106)

Where, the perturbation components, $U = \rho, u, v, w, p$ is described, with respect to the mean flow. The LEE equation is solved using a method called the Quadrature-Free Discontinuous Galerkin spatial discretization approach in which, further references can be obtained from Atkins and Shu (1998). In this method, the LEE equation is multiplied by a weighting function, $W$, and integrated over a finite-dimensional subspace. Thus, the equation is further refined to:

$$\int_{\Omega} W^T L(U_h) d\Omega = \int_{\Omega} W^T S d\Omega$$  \hspace{1cm} (2.107)

Where, $U_h$ and $W_h$ are defined as:

$$U_h = \sum_{i=1}^{N_h} \sum_{k=1}^{M} \nu_{i,k} (t) b_{i,k} (x)$$  \hspace{1cm} (2.108)
\[ W_h = \sum_{i=1}^{N} \sum_{k=0}^{M} w_{i,k} b_{i,k}(x) \]  \hspace{1cm} (2.109)

And \( b_{i,k} \) is the base function, which is further defined as:

\[ \int_{\Omega_i} b_{i,k} L(U_h) d\Omega = \int_{\Omega_i} b_{i,k} S d\Omega \]  \hspace{1cm} (2.110)

In order for the base function, \( b_{i,k} \) to be second order accurate, the equation is modified using Gauss’ theorem and thus written as:

\[ \int_{\Omega_i} b_{i,k} \frac{\partial U_h}{\partial t} d\Omega + \int_{\Gamma_i} b_{i,k} F_j(U_h)n_j d\Gamma - \int_{\Omega_i} \frac{\partial b_{i,k}}{\partial x_j} F_j(U_h) d\Omega = \int_{\Omega_i} b_{i,k} S d\Omega \]  \hspace{1cm} (2.111)

Where, in order to become Quadrature-free, \( S \), is defined as:

\[ S = S_h = \sum_{i=1}^{N} \sum_{k=0}^{3} S_{i,k}(t) b_{i,k}(x) \]  \hspace{1cm} (2.112)
Chapter Three
METHODOLOGY

This chapter describes the methodology for this project. This chapter discuss the general CFD approach and the accuracy associated with it. This is followed by discussion of grid generation techniques, discretization and numerical schemes. This chapter also discusses the near wall model and turbulence models available. Lastly, this chapter describes the CAD model and input boundary conditions used in this project.

3.1 General CFD Approach Process

There are three steps involved in a typical CFD simulation. The three steps are:

- Pre-Processing Stage
- CFD Solving Stage
- Post-Processing Stage

In the pre-processing stage, CFD users are needed to provide sufficient input to the computer in order to obtain the desired output. The pre-processing stage is divided into several steps:

- Geometry Generation
- Mesh Generation
- Input for Boundary Condition
- Flow Type (Steady/Unsteady)
- Discretization Scheme Input
- Turbulence and Near Wall Model Input

The second stage is the solving stage where the physics of the flow is being solved by the computer. The third and final stage is called the post-processing
stage where analysis of graphical representation of the numerical results takes place.

3.2 Accuracy Factors and Errors Associated with CFD

As discussed in previous sections, the limitations of computational performance are hindering CFD in reaching its full potential. Due to these limitations, numerical models are always developed to solve for flow and this leads to assumptions and approximations being made by researchers. According to Naser (1990), numerical models are categorised into two types: the Phenomenological and Multi-dimensional models. Multi-dimensional models are more accurate and extensive since it models both spatial and temporal information of the flow. However, Phenomenological models are much more cost effective since it models only the temporal part of the flow. In general, these numerical models are subjected to solution errors that will affect the accuracy of the CFD results. Numerical solution from CFD has approximately 10 to 15% error comparison to empirical results, Swinburne (2000). The causes of solution errors in a CFD simulation are:

- Error due to Discretization
- Equation solver/Round-off Error

According to Anderson (1995), discretization error is defined as the difference between the exact analytical solutions based on the governing partial differential equation of flow (continuity equation, Navier-Stokes equation together with Turbulence Model that is chosen and Energy equation) and the algebraic equations developed from corresponding numerical scheme. In the discretizing process, the governing partial differential equations is transformed into systems of algebraic equations via numerical scheme, truncation errors (round-off of numerical scheme) are being introduced. Furthermore, the discretization process requires numerical treatment of boundary conditions to take place and this induces
additional errors as well. In addition, the accuracy of the discretization process relies on the grid refinement and quality of the computational domain.

High grids generation and refinement in a given CFD flow domain that results in low discretization error will provide better simulation of the flow physics through better visualization in the post-processing stage. This is especially true for areas within the flow that is subjected to high change in pressure gradients. However, the resulting computation time will take much longer. The refinement procedure is called the grid-independency procedure. In the grid-independency procedure, converged numerical solution is then refined, usually by doubling its density. The flow is then solved again. The grid-independency procedure is stopped once the difference in numerical solution between grid refinements is small, normally within 5.0% of each other. However, it is also important to establish smoothness around the grid refinement areas. Large variations in grid shape or density can cause numerical diffusion, resulting in inaccurate results or simulation instability. In addition, high grid quality also needs to be maintained within the computational domain. There are two factors that determine grid quality. They are:

- Aspect Ratio
- Grid Skewness (Distortion)

Grid aspect ratio is given by the ratio of the length and width ($\Delta y/\Delta x$) of any given grid surface. The ideal aspect ratio is 1.0, with the minimum requirement normally to either be larger than 0.2 or smaller than 5.0, Swinburne (2000). A poor grid aspect ratio will normally lead to poor accuracy in numerical results and a possibility of divergence during solver iterations.

Grid skewness is given by the angle of skewness between two edges on a given grid. The most ideal grid skewness is $\theta=90^\circ$, Swinburne (2000). However, minimum requirement in grid skewness is to be maintained in between $45^\circ$ and $135^\circ$. Poor grid skewness can also lead to low level of accuracy of numerical results and divergence during solver iterations.
According to Anderson (1995), equation solver/round-off error is caused during the solving stage. In the solving stage, iteration process takes place where repetitive calculations are done in order to achieve a numerical solution. The iteration process often “round-off” calculated numbers into significant figures, resulting in error. The iteration process is repeated until the solution converged. Convergence is a limit set by the CFD user to stop the iteration process once the difference between calculated values of the iteration results is too small. Normally convergence limit are set when differences in iteration results are to within 0.1% of each other.

It has been discussed in this section the importance of grid generation and discretization scheme in determining the accuracy of numerical results in CFD simulations. In the next sections, more detailed discussion will be presented on grid generation and discretization schemes in CFD especially on automotive type problem. Discussion will cover various types and implementation strategies used on grid generation and discretization scheme.

### 3.3 CFD Grid Generation and Discretization Methods

In order to generate the highest quality of grids for the highest level of accuracy on a CFD simulation, grids generated depends on the complexity of the flow problem. For high complex flow problems, a more complex high quality grids need to be generated for the CFD simulation. This high level of flow complexity applies for automotive type problems. However, the applications of complex grid generation in automotive flow problems are dependent on the overall performance of the modern day computers. Before high performance computers were developed, a lot of simplifications were made on the governing mathematical equations of the flow. This reduces the complexity of the grids generated for the CFD simulation. The generations of grids and employing of discretization schemes for a simplified flow were done using a Linear or Potential flow method. After the development of supercomputers that are capable in handling complex
turbulence flows, a Non-Linear method was developed for grid generation and discretization schemes.

The Linear methods or Potential flow methods are applied on problems where flow is usually incompressible, inviscid (non-viscous) and irrotational. Governing Navier-Stokes equation is reduced to its linear Laplace form. Flow that uses the linear methods are not practical in solving turbulence flow around a ground vehicle. This is because grid generation for a linear method is usually generated only on the body surface of the ground vehicle geometry. Despite its limitations, the linear methods are still widely used in the industry. However, modelling additional flow phenomena such as vehicle wake and vortex formation requires separate modelling techniques. It does not have the ability to automatically simulate flow separations around the vehicle especially the formation of A-pillar vortices. Further information on the linear methods is available in Ahmed (1998).

In the non-linear methods, the physics of the flow takes into account complex phenomenon such as turbulent properties. Therefore, it requires a much more complex Navier-Stokes equation representation. Since three-dimensionality is now in play, grids generation will occupy the whole computational domain around the geometry of the ground vehicle body. For non-linear methods, the coupling of grid generations and discretization of numerical schemes approximations can be done using either one of three grids discretization methods:

- Finite Difference
- Finite Volume
- Finite Element

The choice of using the finite difference, finite volume or finite element methods depends on the complexity of the flow geometry.

According to Ahmed (1998), the finite difference technique was the first to be developed. The basic idea of finite difference technique is to express the governing partial differential equations approximately into algebraic difference
equations form using finite difference schemes at the grid nodal points. Finite difference methods often employ curvilinear orthogonal grid system that it is difficult and time consuming to generate especially in three-dimensional computational domain. This results in governing transport equations that has more terms. This difficulty makes the finite difference techniques seldom used in automotive CFD.

In the finite volume methods, the computational domain is split up into many small control volumes. The partial differential governing equations are integrated over each of the control volumes and the resulting integrated governing equations are then discretized using finite-difference schemes. A resulting set of algebraic equations is then formed, which are then solved. The main advantage of finite volume methods over the finite difference methods is that the finite volume methods can use both structured orthogonal and non-orthogonal grid system. In the non-orthogonal grid system, irregular unstructured grids can be used. As a result, various grid shapes and size can be used for the finite volume methods. This is however, at the expense of additional computational resource. Because of the flexibility in grid generation of the finite volume methods, it is the most preferable methods used in automobile aerodynamics applications.

In the finite element methods, the computational domain is split up into small volumes, called elements. Within each element, values are being approximated as a linear combination of weighting residuals and use the integral form of the governing differential equations for each element volume without direct reference to other cells. Information is shared among all the other grid points in the element. This method generally uses irregular unstructured grid, often in a triangular shaped element. By employing irregular unstructured grids, the treatment of complex surface geometry is possible and furthermore, it also allows local grid refinements at critical areas without the penalty of simultaneous grid refinements at other areas. However, this also serves as a disadvantage since sufficient refined grids in finite element methods are needed to give solution of high accuracy, Du Pont (2001).
For structured grids, the computational domain is often divided into several blocks. This technique is often defined as Block-Structured or Multi-Block technique. Each block can be constructed with different grid density depending on the flow requirement on that region (i.e. separation, vortex generation). Often in a three-dimensional flow, hexahedral type grid is used for the computational domain. Hexahedral grids have small/no skewness resulting in numerical results of high accuracy. The problem with multi-block structured grids is that most often, the grids do not align up against its adjacent interface of blocks. According to Ahmed (1998), special treatment is needed due to this discontinuity to ensure correct implementation of boundary conditions and to inhibit numerical instability. In addition, with structured grids, grid refinement process is done through the re-mesh and re-calculation approach where the grids in the computational domain is refined from the original coarse model and the CFD simulation is recalculated until a suitable grid independent solution is obtained. This quest in obtaining an independent grid solution process is also known as the grid independency test. The refinement processes for structured grids are such a disadvantage because it is so tedious and time consuming, Makowski et al. (2000).

Grids can also be arranged in an unstructured manner. In three-dimensional flow, grids that are normally arranged in an unstructured manner consist of prisms and tetrahedral grids. Unstructured grids offer flexibility and in especially on area with complex surface geometry. The overall grid generation process is faster with unstructured grids. Grid refinement can be done manually through identifying critical areas within the flow, such as area of separation. Otherwise, grid refinement can be made through automatic solution-adaptive process that identifies large gradients of flow that needs refinements, Makowski et al. (2000), Uchida et al. (1997), Murad et al. (2002). This grid refinement process will basically improve the resolution of flow feature after each computational simulation by reducing steep gradients in the critical areas. However, according to Makowski et al. (2000), the solution adaptive mesh refinement is impractical for time-dependent simulations because the locations of the steep gradients change with time. For an unsteady simulation, accuracy that is comparable to that of a
A well-adapted steady simulation can only be achieved at a much greater total grid count. In addition, solution adaptive mesh refinement is also useful for eliminating oscillatory solution resulted from coarse volume mesh, during a steady flow calculation. However, unstructured grids can sometimes have high value of skewness, reducing the grid quality. This will reduce the accuracy of the numerical results.

Apart from using a blend of prism and tetrahedral grids, a more advance grid generation system called the Arbitrary Cell Technology (ACT) has been developed. A blend of polyhedral unstructured grids is used to mesh and refine complex three-dimensional computational domain. The mesh generation procedure was automatic with very little time consuming manual efforts required. The advantage of ACT is that grids generation, refinement and manipulation can be done with extremely ease while retaining the smoothness of the complex geometry in the computational domain. However, grid refinement process is done through the same process as structured grids by using the remesh and recalculation approach each time a new refinement is needed.

A combination of hexahedral/tetrahedral hybrid grids can also be used in the computational domain (Murad, 2004). Flow close to the vehicle body can be treated with structured highly accurate hexahedral mesh and free stream flow far away from the body can be treated using unstructured tetrahedral mesh. Flow treated using structured hexahedral mesh was done using multi-block method. The major setback to this technique is that it takes a long time to generate. Grid refinement process is also hard to do since grid refinement process through solution adaptive process will upset the whole balance of the generated grids structure. This technique is used in a computational domain where a final adapted grid count and grid refinement location has been established.

In order to estimate the size of the time step require to solve for a given grid interval, the Courant number relationship can be used. The Courant number, $\text{Cr}$ is defined as:
which is the ratio between the product of flow velocity and time step and grid interval.

\[ Cr = \frac{v \Delta t}{\Delta x} \]  

(3.1)

3.4 CFD Numerical Schemes

Apart from grid generation, the choice of numerical or interpolation scheme in discretization methods implementation are important to ensure numerical solution of high accuracy in order to limit discretization error. Once the computational domain has been meshed, a set of procedure takes place in discretizing the governing equations of flow into its linear algebraic form. In CFD, this procedure is called the discretization or numerical methods. The discretization of governing equations can be done two ways, either by using a segregated or a coupled solver. This section discusses the difference between two solvers. In addition, numerical schemes used to discretize each equation are also discuss together with the problems associated with each numerical scheme and the treatment used to overcome the problems.

A good numerical scheme should be able to provide fast convergence of the iterative solution. This can be achieved by having a numerical scheme that is stable and consistent. The numerical scheme is said to be stable by being bounded during the iteration process (does not diverge and oscillate). Consistency of the numerical scheme is achieved through the ability of obtaining the real solution (reduction of the truncation error) through grid independency test. The factor that determines the suitability of various numerical schemes on a particular problem is the Peclet number. The Peclet number is defined as:

\[ \text{PecletNumber} = \frac{\text{ConvectionStrength}}{\text{DiffusionConduction}} = \frac{\rho u \Delta x}{\Gamma} = \frac{F}{D} \]  

(3.2)
For discretization of the momentum equation, the most basic numerical scheme that can be used is the central difference scheme. The central difference scheme is a higher order scheme. However, the central difference scheme can lead to unrealistic results especially for Peclet number that is higher than 2. According to AVL (2003), central difference scheme can generate numerical oscillations yielding unbounded and non-monotonic solutions. The central difference scheme is used for low Reynolds number flow. For flow with low Peclet number, the central differencing scheme is recommended.

For flow with high Peclet number an alternative substitution to the central difference scheme is to use the upwind scheme. A standard upwind scheme is a lower order scheme. In the upwind scheme, the value of velocity for the momentum at the control volume interface is taken as equal to the value of the grid point on the upstream side of the interface (Patankar, 1980). However, according to Patankar, at very high Peclet number, the upwind numerical scheme will yield results that lead to false diffusion. False diffusion will occur for multidimensional problem especially with flow that is not aligned with the grids. A remedy recommended by Patankar is to use refined grids at area that produce false diffusion. A higher order upwind scheme or its modified version (QUICK numerical scheme) can also be used to reduce false diffusion by taking into account multidimensional problem (Fluent, 2003, Swinburne, 2000). However, the QUICK scheme is suited mostly for problems generated using structured grids.

Apart from the central differencing and upwind scheme, standard numerical scheme that are available on most CFD software are exponential scheme, hybrid scheme and power law scheme. Although this numerical scheme does not still solve the problem of false diffusion, the development and appreciation of such schemes are still worth discussing. The exponential scheme is developed from the exact solution of the convection-diffusion term. From the exact solution, it was discovered that the Peclet number behaves exponentially especially at large values. This led to the false diffusion problem encountered in the lower order upwind scheme. The exact solution solved this problem by making the coefficient
of the linear discretized equation to match the exponential behaviour. However, the shortcoming of the exponential scheme is that it is expensive to compute and still does not provide an accurate solution for multidimensional problem. This led to the development of the hybrid and Power Law Scheme. Both of these schemes are of lower order. The hybrid scheme improves the exponential scheme by making it more economical to use. This was implemented by using the central differencing scheme at Peclet number between 2.0 and –2.0. At Peclet number higher than 2.0 or lower than –2.0, the upwind scheme is used. The Power-Law scheme further refine the hybrid scheme to make it more accurate without sacrificing much computational resource by combining the exponential and the upwind scheme. The concept is similar to the hybrid scheme.

In addition, AVL (2003) introduces two higher numerical schemes based on the ‘Total Variation Diminishing Concept (TVD)’ and the ‘Convection Boundedness Criterion (CBC)’ of Sweby (1984) and Gaskell et al. (1988). These higher order numerical schemes are known as the MINMOD and SMART schemes respectively. The MINMOD schemes combine the lower upwind differencing scheme and the central differencing scheme while the SMART scheme coincides with the QUICK scheme. The MINMOD scheme offers better convergence properties than the SMART scheme, which means that it is more diffusive (less accurate) on steady state cases.

From the momentum equation, in order to solve for the velocity values, the pressure values needs to be known. Unfortunately this is not given and a guessed value for the pressure is substituted in the equation. For structured grids, a staggered grid procedure needs to be applied during discretization for the momentum equation. This is to prevent wavy or checker board effect for the velocity (in the continuity equation) and pressure term (in the momentum equation) (Patankar, 1980). In the staggered grid procedure, the velocity components are calculated from the points that lie on the faces of the dotted control volumes. The pressures on the other hand are calculated based on the central node points on the shaded control volume.
Once the guessed pressure has been substituted into the discretized momentum equation, a guessed velocity value will be obtained. Ultimately, the objective of the exercise is to obtain the correct value for velocity and pressure. In order to do this, the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used. These values for the correct velocity and pressure are given as:

\[ P = p^* + p' \] (Correct Pressure = Guessed Pressure + Corrected Pressure)

\[ u = u^* + u' \] (Correct Velocity = Guessed Velocity + Corrected Velocity)

In order to obtain the values for the pressure and velocity correction, the real and guessed values from the momentum equation is first interpolated and substituted, semi implicitly into the continuity equation to obtain values for \( p' \). \( u' \) is then obtained back from the resulting interpolated momentum equation. Once the correct pressure and velocity values are obtained, it is then stand for the new guessed values for both velocity and pressure. The procedure is then repeated again until a converged solution is obtained. The main shortcoming of the SIMPLE algorithm is the slow convergence rate. This is due to the omission of the neighbourhood term (\( \sum a_{ab}u_{ab} \)) in the momentum equation interpolation. The omission of the neighbourhood term will cause an exaggeration in the pressure correction value, \( p' \). This can cause divergence in the calculation and to overcome this, under relaxation has to be introduced, which will lead to a slow convergence rate.

As mentioned before, the staggered grid approach is used for computational domain with structured grids. Automotive type application however, uses mostly unstructured grids and this requires a slightly different approach for the discretized momentum and continuity equation to solve for velocity and pressure. Patankar (1980) explained a staggered grid approach for unstructured grids. In the approach, the pressure value is calculated in the main node point that completes the unstructured grid. The unstructured grid is then further divided into several sub-grids. Velocity and other variable values are then calculated from the nodes of those sub-grids. Another approach for unstructured grids is called the collocated
grids (Peric, 1999). Unlike the staggered grids approach, the collocated grids approach calculates and stores all the variable values in the same grids point. With this approach, the SIMPLE algorithm used will be slightly different, Rhie et al. (1983). The values of pressures and velocities on the face of the control volume will need to be interpolated from values stored in the grid nodes. The beauty of the collocated grids approach is that since all the variable values are stored in the grid points, the shape of the grids can be of any size and form.

3.5 Segregated and Coupled Solver

The segregated solver and the coupled solver both employ finite volume discretization methods. However, the process used to discretize and solve the resulting linear algebraic equations is different. In the segregated solver, the governing equations are first discretized and then solved sequentially, or separately from each other. By using the initial boundary condition values, the discretized momentum equation is solved first in order to update the velocities values. The pressure value is then updated by solving the continuity equation locally, using a pressure-velocity coupling treatment. Lastly, the equation for scalars values such as turbulence is then solved. Several iterations must be performed until a converged solution is obtained.

The coupled solver solves the governing equations of momentum and the continuity equation simultaneously. The governing scalar equations are solved separately i.e. segregated from one another. Again, the steps are repeated for several iterations until a converged solution is obtained.

In the case for automotive flow problems where the flow is incompressible and subsonic, the segregated solver is used (CFD-online, 2002). Furthermore, since most available CFD solver adopts the segregated method of solving for the governing mathematical equations, it will be focused on and discussed in this section.
This chapter describes the CFD pre-processing procedures for this project such as the creation of CAD models, methods of grid generation and boundary conditions used. In addition, this chapter also presents the most feasible grid generation method used for simplified vehicle model with different windshield radii. This is done through comparison of various grid generation methods.

3.6 Near Wall Models and Turbulence Models


Launder and Spalding (1974) developed the standard wall function used in many CFD applications used today. The development of the standard wall function was based on combining Boussinesq assumptions with the Prandtl-Kolmogorov relation for turbulent eddy viscosity in achieving the universal logarithmic law of the wall in obtaining relationship for the mean velocity in the near wall region. The calculation of mean velocity, together with the kinetic energy and dissipation budget near to the wall will be based on the centre point of the cell adjacent to the wall. The advantage of the standard wall function is to a certain extend provide a robust and rudimentary solution to the near wall behaviour without requiring extensive mesh refinements.

Chieng and Launder (1980) was the first to introduce the concept of two-layer wall modelling in which the limit for the wall shear stress was set based on the turbulent kinetic energy budget at the edge of the viscous sub-layer region. The turbulent kinetic energy budget was determined through simultaneously modelling the thickness and Reynolds number in the viscous sub-layer region. The main benefit of using the wall treatment model was bypassing the need for mesh integration right down throughout the viscous sub-layer region. Kim et al. (1995)
showed that under adverse pressure gradient conditions, the two-layer wall model yielded an improved performance compared to the standard wall function of Launder and Spalding (1974).

The model of Lam and Bremhorst (1981) is a low Reynolds number model developed to complement the standard $k$-$\varepsilon$ turbulence model in the near wall region. The integration of this wall treatment model requires grid refinement all the way to the viscous sub-layer region of the wall. This model includes improvement to the damping function in the $\varepsilon$ transport equation to account for the change of turbulence length scale in the near wall region.

The enhanced wall treatment uses the Wolfstein (1969) one equation model in the low Reynolds number, viscous affected region. The modified version of the Wolfstein (1969) model (FLUENT, 2003) uses a blending formulation in modelling for the turbulent eddy viscosity in an effort to smooth the transition between the fully turbulence and viscosity affected sub-layer region. The length scale for the diffusion and dissipation term in the one equation model are also assigned with damping factors to better model the viscosity affected region. The one equation model has shown to offer good prediction in near wall flows due to its empirical nature in providing a turbulence length scale to the flow, Wilcox (2002).

The final wall treatment model that was chosen in this project was the Wall Elliptic Blend model of Manceau and Hanjalic (2002) to complement the near wall approach modelling strategy used together with the RSM of LRR and SSG. The near wall treatment model was developed to satisfy the universal constraints of turbulence properties close to the wall while at the same time maintaining the simplistic approach to the numerical complexities of the model. This was done for the main purpose of offering practicality for industrial applications. This was achieved through modifying and simplifying the redistribution term in the Reynolds Stress Near wall model of Durbin (1993) by using an elliptic blending function.
The selections of turbulence models that were chosen for this project is the standard $k$-$\varepsilon$, the $k$-$\varepsilon$ RNG (Re-Normalization Group), Realizable $k$-$\varepsilon$ and the $k$-$\omega$, Reynolds Stress Model (RSM) and the Hybrid Turbulence Model (HTM2).

The standard $k - \varepsilon$ turbulence model of Launder and Spalding (1972), is the standard turbulence model used in almost every CFD study as a baseline for comparison with other turbulence model and will be chosen for this study. Due to the use of Boussinesq postulations and eddy viscosity formulation for the modelling of Reynolds stresses, the standard $k - \varepsilon$ turbulence model is more suited to modelling high Reynolds number flows type application, in which the turbulence behaviour are more isotropic in nature. The stable and robust characteristics of the standard $k - \varepsilon$ turbulence model made it a popular choice particularly for industrial application. Its deficiencies in modelling low Reynolds number flow near to the wall and flow subjected to pressure gradient effects have since been addressed by other researchers. Such effort to improve the standard $k - \varepsilon$ turbulence model includes the work of Yakhot and Orszag (1986) and Shih, Liou, Shabbir, Yang and Zhu (1995).

Yakhot and Orszag (1986) conducted a study through the use of Re-Normalization Group theory (RNG) in improving the standard $k - \varepsilon$ turbulence model. Improvements included the modification of the diffusion term in the $k$ and $\varepsilon$ transport equation enabling it to perform better in near wall, separating and swirling flows. In addition, an extra source term was introduced in the $\varepsilon$ transport equation that improves prediction of rapidly strained flows and the effect of streamline curvature, FLUENT (2003). Validation of the RNG $k - \varepsilon$ model by Lien et al. (1994), Kim et al. (1995) and Wilcox (2002) showed an improve performance over the standard $k - \varepsilon$ model especially in adverse pressure gradients flow.

Shih, Liou, Shabbir, Yang and Zhu (1995) introduced the “realizable” $k - \varepsilon$ turbulence model to address the non-physical turbulence characteristics displayed by the standard and RNG $k - \varepsilon$ turbulence model that occurred due to use of a constant value of $C_\mu$ in modelling for the turbulent eddy viscosity. By using a
variable $C_\mu$ to better model the Reynolds stresses in flow situated close and far away from the wall. The realizable $k-\varepsilon$ also introduced a modified version of the $\varepsilon$ transport equation to better model the spreading rate of planar and round jet in free shear flow, FLUENT (2003). The realizable $k-\varepsilon$ turbulence model have shown to perform better than the standard $k-\varepsilon$ turbulence model in modelling turbulent boundary layer flows, subjected to adverse pressure gradient effect that includes separation and recirculation, Shih et al. (1995).

Wilcox (1998) developed the $k-\omega$ turbulence model in an attempt to provide a superior choice in two-equation turbulence model over the $k-\varepsilon$ turbulence model family. By means of the perturbation method, Wilcox (2002) diagnosed the $k-\omega$ model to reveal that the superior performance over the traditional $k-\varepsilon$ turbulence models was due to its ability to model the defect region within the boundary layer. According to Wilcox (2002), successfully modelling the defect layer will consequently result in better prediction of the transition and viscous sub-layer region of the boundary layer. Validation by Wilcox (2002) for free shear flow application showed better performance in comparison to the standard and RNG $k-\varepsilon$ turbulence model. The $k-\omega$ turbulence model was developed to perform down to the viscous sub-layer region to account for low Reynolds number effects close to the wall, requiring mesh refinement down to the viscous sub-layer region of the wall.

The Hybrid Turbulence Model (HTM2) of Basara and Jakirlic (2002) was developed behind the idea that the $C_\mu$ coefficient used to determine the turbulent eddy viscosity varies across a wide range of turbulent flow and should remain constant. Instead of using a standard constant for $C_\mu$, the $C_\mu$ coefficient is taken as a ratio between the production of turbulence kinetic energy of a Reynolds Stress Model and a standard $k-\varepsilon$ model. The variable values of $C_\mu$ suggested that the HTM2 turbulence model should perform with the robustness of the standard $k-\varepsilon$ model while providing accuracy in results similar to the RSM turbulence model, Basara et al. (2002).
The Reynolds stress turbulence model (RSM) used in AVL SWIFT CFD is of Launder Reece and Rodi (LRR) (1975). The redistribution term used was that developed by Speziale, Sarkar and Gatski (SSG) (1991). The RSM of LRR is the most widely used second moment closure turbulence model. Its advantages include improvement over the eddy viscosity models in predicting complex three dimensional flow that are subjected to curved surfaces, swirl, rotation, rapid variation of strain rate and boundary layer separation, FLUENT (2003), Wilcox (2002). These are implemented by modelling Reynolds stresses via transport equations, by passing the traditional method of the Eddy Viscosity method. The combination of LRR and SSG as an RSM provides a superior performance in a variety of complex flow over the standard LRR model, Speziale et al. (1991) and Basara et al. (2003).

3.7 CAD Model Geometry and Boundary Conditions

Input

The CFD models used for this project were based on five different simplified vehicle model (40% scale) of Alam (2000). The models are different form each other in terms of their A-pillar windshield radius configuration. Alam (2000) defined the five simplified vehicle models as:

- Rectangular Edge (RE)
- Slanted Edge (SL)
- Semi Circular (Semi)
- Small Ellipsoidal (SE)
- Large Ellipsoidal (LE)
It can be seen from Figure 3.1 that the rectangular edge model does not have any slant angle. All the other models have a slant angle of 60° from the $y$-axis component, which is the slant angle among production vehicles Alam, 2000. In addition, among all the models, the windshield of the RE and SL models does not have any radius of curvature. The Semi, SE and LE models have windshield curvature radii of 374, 299 and 449 mm respectively. Furthermore, all five models were configured in the computational domain to simulate the effect of cross wind. With this, all models were positioned to yaw 0°, 5°, 10° and 15° respectively (Figures 3.2, 3.3 and 3.4).
Figure 3.2: Models in 0°, 5°, 10° and 15° yaw position

Figure 3.3: Small Ellipsoidal Model in a yaw position within Computational Domain in AVL
All five models were given a length of the bottom 1963 mm, a width of 748 mm and height of 300 mm. The canopy or top section of the models are 288 mm in dimension and contains two rows of pressure tapping points, which are 96 mm from each other. For the RE and SL model, the first pressure tapping point is situated 384 mm from the centre of the windshield or 10 mm from the edge of the A-pillar. For the Semi, SE and LE models, the first pressure tapping point is situated at 490, 470 and 530 mm from the centre of the windshield respectively. Each row consists of 16 pressure-tapping points and each point is 32 mm apart from each other (Figure 3.1).

The computational domain that surrounds the CFD model is generated base on the dimension of the RMIT University wind tunnel in Melbourne, Australia. The RMIT wind tunnel dimensions were measured at 9.0 metres in length, 3.0 metres in width and 2.0 metres in height.

The computational domain and the simplified vehicle CFD models were generated to be simulated using FLUENT and AVL. For FLUENT, a geometry and mesh builder called GAMBIT (version 1.3) was used, which is also a product developed by FLUENT Incorporated. CAD models generated in GAMBIT were saved in a DBS file before being used for grid generation. For AVL, AUTOCAD was used to generate all the CAD models. The CAD models are then saved in an STL file before being exported to AVL for grid generation. The CAD models generated for FLUENT and AVL can be seen in Figure 3.5 and 3.6 respectively.
Before a CFD simulation can be conducted, appropriate boundary condition values must first be specified to the CFD solver. Three separate set of simulation was carried out at vehicle speeds of 60, 100 and 140 km/h respectively. The boundary condition values were specified at the inlet face of the computational domain velocity inlet. This correspond to Reynolds number of $2.169 \times 10^6$, $3.615 \times 10^6$ and $5.061 \times 10^6$ respectively. The boundary condition at the outlet face of the computational domain was set as pressure outlet.
In FLUENT, the convection scheme and the pressure-velocity coupling used for the simulation was set as first and second order upwind differencing scheme and SIMPLE scheme respectively. Default under-relaxation values were used. The convergence level for the residuals for the simulation was set at 0.1%. Throughout the calculation, under-relaxation values were reduced whenever solution showed instability and divergence. The reduction of under-relaxation values varies from one simulation to the other and depends on such factors as inlet velocity, convection scheme used and turbulence model used.

In AVL, the simulation was first carried out using first order upwind scheme and central differencing scheme. Once convergence was reached, the AVL Smart Bound higher order scheme was then used. This was done by progressively altering the blend factor constant in the differencing scheme option. For the higher order scheme was used with a 0.5 blend factor. The convergence level was set to 0.1% with SIMPLE used as the pressure-velocity coupling scheme. The reduction of under-relaxation values also varies from one simulation to the other.

For the turbulence models used, initial guess values were specified for the turbulence kinetic energy \( k \) and dissipation rate \( \varepsilon \), which was determined from the formula of turbulence intensity and dissipation length scale. The turbulent intensity was taken as 1.8% (based on Alam, 2000) and the dissipation length scale was based from 1.0% of the simplified vehicle model height, which resulted in 5.88 mm. These equations are defined as:

\[
I = \frac{2k}{3v_{inlet}}, I_{\varepsilon} = \frac{C_{\mu}k^{3/2}}{\varepsilon}
\]

(3.3)

For the three inlet velocities, 60, 100 an140 km/h, this resulted in the guessed values of 0.135, 0.375 and 0.735 for \( k \) and the values of 0.759, 3.515 and 9.645 for \( \varepsilon \).
The summary of the boundary conditions, numerical scheme along with the turbulence and near wall model used for FLUENT and AVL are listed in Table 3.1 and 3.2.

Table 3.1: Boundary conditions, numerical schemes, turbulence and near wall model for FLUENT

| **Inlet Boundary Conditions** | Velocity Inlet at 60, 100 and 140 km/h |
| **Outlet Boundary Conditions** | Pressure Outlet |
| **Yaw Angles** | 0°, 5°, 10° and 15° |
| **Convection Scheme** | First order and Second order upwind convection scheme |
| **Pressure-Velocity Coupling Scheme** | SIMPLE |
| **Turbulence Model** | Standard $k$-$\varepsilon$ (initial), $k$-$\varepsilon$ RNG (Re-Normalization Group), Realizable $k$-$\varepsilon$ and $k$-$\omega$ |
| **Near Wall Model** | Hybrid Grids with One Equation Model of Wolfstein (1969) |
| **Convergence Level** | 0.1% |
| **Turbulence Intensity Used** | 1.8% |
| **Length Scale Used** | 5.8mm (1.0% of vehicle height) |
Table 3.2: Boundary conditions, numerical schemes, turbulence and near wall model for AVL

<table>
<thead>
<tr>
<th>Inlet Boundary Conditions</th>
<th>Velocity Inlet at 60, 100 and 140 km/h</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Boundary Conditions</td>
<td>Pressure Outlet</td>
</tr>
<tr>
<td>Yaw Angles</td>
<td>0º, 5º, 10º and 15º degrees</td>
</tr>
<tr>
<td>Convection Scheme</td>
<td>Central Differencing and AVL Smart Bound convection scheme</td>
</tr>
<tr>
<td>Pressure-Velocity Coupling Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Standard $k$-$\varepsilon$, Reynolds Stress Model (RSM) and the Hybrid Turbulence Model (HTM2)</td>
</tr>
<tr>
<td>Convergence Level</td>
<td>0.1%</td>
</tr>
<tr>
<td>Turbulence Intensity Used</td>
<td>1.8%</td>
</tr>
<tr>
<td>Length Scale Used</td>
<td>5.8mm (1.0% of vehicle height)</td>
</tr>
</tbody>
</table>
Chapter Four

RANS CFD SIMULATION OF A-PILLAR AERODYNAMICS

This chapter is divided into two main sections. The first section presents the CFD model development for this project. The second section presents the results and discussion of the A-pillar aerodynamics obtained from the CFD model developed. Validation of the CFD results against the experimental data are also presented in the second section.

4.1 Objective and Scope of this Chapter

From the knowledge obtained while researching for this project through reviews of literatures, the following questions were raised and will be addressed in this chapter.

- What are the most suitable combination of grid, near wall model and turbulence model to use for the CFD model in the project?
- How accurate are the proposed turbulence and near wall model in terms of percentage error values? Are the errors within acceptable values?
- Using flow visualisation, what are the physical characteristics and mechanism of vortices generated from the A-pillar aerodynamics?
- Using flow visualisation, how do different vehicle windshield radii configurations affect A-pillar aerodynamics together with the vortices and turbulence generation associated with it?
Therefore, the main objectives of this chapter are twofold:

1. To develop a CFD model for the project by obtaining the best combination of grid, near wall model and turbulence model for the various windshield radii and yaw angle. CFD model results were validated against the existing experimental data of Alam (2000).

2. To examine the aerodynamics behaviour behind the A-pillar region using the developed CFD model by means of elucidating the mechanism of turbulence generation and airflow physical characteristics from the various flow conditions through further justifications with relevant works of literatures.

4.2 CFD Model Development

Development of the CFD model will be discussed in this section. This includes the combination for the most feasible grid, near wall model and turbulence model.

4.2.1 Grid Feasibility Study – Generation Technique

Different grid generation methods were used to mesh the CFD models in both GAMBIT and AVL. The grid generation methods differ from each other based on two factors, the difficulty of the method and also on the accuracy that each method provides. The grid generation methods used were categorised into either a conformal or non-conformal generation. The conformal grid generation technique used comprised of either using structured or unstructured grids. The non-conformal grid generation technique used comprised of using the multiblock grid technique.

In GAMBIT, both the non-conformal and conformal grid generation techniques were implemented. For this project, the non-conformal multiblock grid generation
technique was constructed using fully hexahedral grids and hybrid grids. The conformal grid generation technique was constructed using fully unstructured tetrahedral grids.

The non-conformal multiblock grid generation technique used in GAMBIT was the most difficult to implement. They can be constructed using fully hexahedral grids or a combination of hexahedral, prismatic and tetrahedral grids, which combination are often known as hybrid grids. In the multiblock technique, the entire computational domain can be divide into several volume blocks. Each adjacent blocks are connected via an interface boundary condition with each block meshed independently. Generating grids using the multiblock techniques can be a problem in GAMBIT because the interface boundary conditions might fails to connect if the adjacent blocks are too big. In the case for the multiblock technique, smoothing technique must also be applied between adjacent blocks in order to prevent numerical instability. The slanted edge model implemented with the multi block grid generation technique can be seen in Figure 4.1.

![Figure 4.1: Slanted Edge Model with Multi-block volumes in GAMBIT](image)

The fully structured hexahedral grids generated by using the multi block technique possess the highest grid quality due to its conformity with the aspect
ratio and skewness angle requirement. Mesh analysis from GAMBIT showed that only 0.05% grids are outside the aspect ratio requirement and 0.8% grids are outside the skewness angle requirement respectively. This should minimise discretization error and most often than not, translate to accurate CFD results. However, the generation of hexahedral grids throughout the computational domain was an expensive exercise. The structured arrangement of the hexahedral grids required fine grids to be generated in areas within the computational domain, often located in the non-critical flow region. This is inappropriate since maximum accuracy is only needed for this project on the area surrounding the A-pillar region. Therefore, an alternative to using the fully structured hexahedral grids is to use hybrid grids in order to obtain a more efficient grid distribution throughout the computational domain. The fully structured hexahedral grids layout can be seen in Figure 4.2 with the mesh quality can be seen in Figure 4.3.

![Figure 4.2: Fully Structured Hexahedral Grids Layout in GAMBIT](image1)

![Figure 4.3: Grid Skewness Quality of the Fully Structured Hexahedral Grids in GAMBIT](image2)
The hybrid grids generated using the multiblock technique are a combination of hexahedral, prismatic and tetrahedral grids. The area surrounding the surface of the vehicle model was meshed using hexahedral grids while the surrounding area was meshed using tetrahedral grids. Prismatic grids were used to connect the hexahedral and tetrahedral grids respectively. The reason to generate hexahedral grids surrounding the surface of the model was to obtain maximum accuracy, particularly around the A-pillar region. Furthermore, hexahedral grids are also needed near the surrounding model surface in order to successfully generate fine boundary layer mesh for simulating laminar flow in the sub-layer region. Everywhere else within the computational domain, tetrahedral grids were generated. Due to its unstructured arrangement, less grids were used to generate tetrahedral grids in which, reasonable accuracy can be obtained. Mesh analysis from GAMBIT showed that 24.6% grids are outside the aspect ratio requirement and 20.7% grids are outside the skewness angle requirement respectively. The reason for the high number of increase in aspect ratio percentage is due to the generation of boundary layer close to the model surface. The high increase of skewness angle however, is due to the generation of tetrahedral grids outside the critical of the flow. The hybrid grids layout can be seen in Figure 4.4 with the mesh quality can be seen in Figure 4.5.

Figure 4.4: Hybrid Grids Layout in GAMBIT
In the conformal grid generation technique used in GAMBIT, fully unstructured tetrahedral grids were used. They are the easiest to generate in comparison to the fully structured hexahedral and hybrid grids. However, the unstructured tetrahedral grids are low in quality since it has high skewness and aspect ratio. This might result in high discretization error, especially when exposed to A-pillar vortex flow. Mesh analysis from GAMBIT showed that none of the grids are outside the aspect ratio requirement and 27.0% grids are outside the skewness angle requirement respectively. The reason for the low percentage of aspect ratio is due to the generation of purely tetrahedral grids. The percentage of skewness angle is similar as per the hybrid method in which is caused by the generation of tetrahedral grids in the computational domain. The unstructured tetrahedral grids layout can be seen in Figure 4.6 with the mesh quality can be seen in Figure 4.7.
In AVL, a built in grid generator tool call Fame Hybrid was used. Fame Hybrid uses a more advance meshing system called the Arbitrary Cell Technology (ACT). In the ACT meshing system, a blend of conformal polyhedral grids can be used to mesh and refine complex 3-D CFD models. The grid generation procedure can be done either manually or by using an automatic grid generator. In comparison to GAMBIT, highly complex grids can be generated easier by using Fame Hybrid and without having to use the multiblock technique. The resulting mesh quality is probably similar to those of Hybrid grids from Gambit. The polyhedral grids layout from Fame Hybrid can be seen in Figure 4.8.

Figure 4.7: Grid Skewness Quality of the Unstructured Tetrahedral Grids in GAMBIT
For this project, the non-conformal fully structured hexahedral in GAMBIT were implemented only on the sharp edge models (RE and SL) at 0° yaw due to its tedious and difficult nature of generation. The non-conformal hybrid grids and conformal unstructured tetrahedral grids in GAMBIT were implemented to the sharp edge models (RE and SL), along with the circular models (Semi, SE and LE) at 0°, 5°, 10° and 15° yaw. In AVL, polyhedral grids generated by using Fame Hybrid was applied to the sharp edge (RE and SL) and circular models (Semi, SE and LE) at 0°, 5°, 10° and 15° yaw.

Implementation of the various grid generation techniques shows that the tetrahedral and the AVL polyhedral grids are the easiest to generate among all the grid generation methods used. In the next section, we shall look on the accuracy and the applicability of these grid generation methods.
A summary listing the grid generation techniques used for both the models in GAMBIT and AVL Fame Hybrid are as follows:

- **GAMBIT**
  - Non-Conformal Hexahedral – Sharp Edge models at 0° yaw.
  - Conformal Tetrahedral – Sharp Edge and Circular models at 0°, 5°, 10° and 15° yaw.
  - Non-Conformal Hybrid – Sharp Edge and Circular Models at 0°, 5°, 10° and 15° yaw.

- **AVL Fame Hybrid**
  - Polyhedral ACT – Sharp Edge and Circular models at 0°, 5°, 10° and 15° yaw.

### 4.2.2 Grid Feasibility Study – Grid Refinement & Independency Procedure

Initial grids generated in GAMBIT and AVL Fame Hybrid are coarse. The coarse grids were then subjected to a grid independency test. In the grid independency test, grids were repeatedly refined especially in areas with high variable flow gradients until less than 5.0% in error was achieved relative to the results from previous refinement. For this project, only models subjected to yaw of 0° and 15° were used for the grid independency testing, to investigate least and worse case airflow scenario.

For the grid independency test, the CFD models generated using the non-conformal and conformal grid method in GAMBIT were subjected to a wind tunnel inlet velocity of 60 km/h. Standard $k$-$\epsilon$ turbulence model was used for the simulation. For models meshed using the non-conformal grid method (hexahedral and hybrid), the modeling was done in two scenarios, with and without boundary layer grids added to the wall. In the scenario with boundary layer grids, near wall conditions for the grid independency test for FLUENT was conducted using
Wolfstein (1967, 1969) one equation model. In the scenario without the boundary layer grids, and this also applies to the models meshed with conformal unstructured grid method, wall conditions were of the non-equilibrium wall function model.

For AVL Fame Hybrid, all CFD models were subjected to an inlet velocity of 60 km/h. Standard $k$-$\varepsilon$ turbulence model with standard near wall function model was used. Boundary layer grids were later added to the sharp edge models at 0° yaw and for all models at 15° yaw. Near wall conditions for the grid independency test was then conducted using Lam et al. (1981) low Reynolds number model.

In GAMBIT, the grid refinement for the conformal unstructured grids was made through automatic solution adaptation technique. For the non-conformal hexahedral and hybrid grids, the final amount and combination of grids generated was based on the grid refinement obtained from the grid independency test done on the polyhedral grids (which is similar to hybrid grids) in AVL.

The initial surface grid size created on the vehicle models wall surface (before refinements) for the unstructured tetrahedral in GAMBIT was 20 mm in size while the surface grid on the tunnel wall was 200 mm in size. For the hexahedral and hybrid grids, the final grids generated on the vehicle wall surface was 20 mm with surface grids surrounding the A-pillar region generated at 5 mm in size. For models constructed using the fully unstructured tetrahedral grid method, the total initial grids generated was around 250,000. The final grid total after achieving grid independency was around 500,000 (Refer to Figure 4.9). For models generated using the hexahedral and hybrid grid method, the final grids generated was around 1.0 million with 477,000 grids generated on each side of the A-pillar region.
For the hybrid grid strategy, boundary layer mesh was added to better capture the airflow in the boundary layer region in order to obtain a more accurate static pressure distribution on the vehicle surface. The first boundary layer mesh point was at 0.03 mm perpendicular to the vehicle surface resulting in the $y^+$ value close to the wall to be close to 1. A total of 14 boundary layer mesh was generated in 1.4 ratio increments resulting to a total depth of 8.3mm. The final grids generated along with added boundary layer grids for the sharp edge model totalled to approximately 1.4 million (Refer to Figure 4.10)
In AVL, an initial coarse wall surface grids of 200 mm in size was generated on the wind tunnel wall. For all models, an initial surface mesh of 100 mm in size was generated on the vehicle model surface. The wall surface mesh was later refined to 50 mm, 25 mm and 12.5 mm in size respectively. However, on the slanted edge model at 0° yaw and for all models at 15° yaw, a further refinement was conducted on the wall surface surrounding the A-pillar region with generated grid size of 5.0 mm in size. In addition, a choice of 5 and 10 boundary layer grids were constructed on the models surfaces with the first grid next to the wall measuring 0.1 and 0.01 mm in size respectively. The $y^+$ value close to the wall obtained after the addition of boundary layer grids was less than 1. Grid independency test for AVL was conducted using the re-meshing technique. Using this technique, new sets of refined grids were generated after a converged solution were obtained from previous set of grids and repeated until error of results were restricted to a maximum of 5.0% relative to previous refinement results. The initial coarse grids generated using Fame Hybrid for the models at 0° yaw was around 20,000, which was far less than the total grid count from GAMBIT. The final grid count after grid independency test for the circular models was around 200,000 grids (Refer Figure 4.11). For the sharp edge model at 0° yaw and all models at 15° yaw, the final grid refinement was done at around 1.7 million grids with minimum improvement. At this point, a combination of either 5 or 10
boundary layer grids were added to the fourth and fifth grid refinement in a different approach to obtain grid independency (Refer Figure 4.12).

Figure 4.11: Grid Independency Test for the Semi Circular Model in AVL at 0° Yaw
Figure 4.12: Grid Independency Test for the Semi Circular Model in AVL at 15° Yaw

A summary listing the initial and final grid refinement used for both the models in GAMBIT and AVL Fame Hybrid through the grid independency testing procedure are as follows:

- **GAMBIT**
  - Fully Unstructured Tetrahedral Grids generated on the Circular and Sharp Edge models – Initial 250,000; Final 500,000.
  - Fully Hexahedral and Hybrid Grids generated on the Circular and Sharp Edge models – Initial 250,000; Final 1 million (without boundary layer), 1.4 million (with boundary layer).
4.2.3 Grid Feasibility Study – Validation with Experimental Results at 0° Yaw

The grid independency test results obtained for the circular models at 0° yaw compared against the experimental results of Alam (2000). From Figures 4.13 to 4.14, it can be seen that close agreements were obtained with experimental results for models simulated using FLUENT and AVL SWIFT. However, a more consistent prediction was obtained using AVL SWIFT. From the results obtained from FLUENT and AVL SWIFT, the most feasible grid generation method to be implemented for future CFD modelling of the circular models at 0° yaw are by using the Fame Hybrid polyhedral grids with standard wall function. Apart from the overall consistency that was obtained by using this method, this strategy was also chosen because it is easy to generate and uses only around 200,000 grids, which was almost half the total final grid count to FLUENT (around 500,000).
Figure 4.13: Grid Independency Test Results for the Semi Circular Model in FLUENT at 0° Yaw (br – bottom row at 250, 000, 500, 000 and experimental at 60 km/h)

Figure 4.14: Grid Independency Test Results for the Semi Circular Model in AVL at 0° Yaw (br – bottom row at 28, 000, 46, 000, 100, 000, 180, 000 and experimental at 60 km/h)
The results from the grid refinements of the sharp edge models at 0° yaw were compared to the experimental results of Alam (2000). From Table 4.1 and Figures 4.15, 4.16 and 4.17, it can be seen that among the grid generation methods used in FLUENT, the Hybrid grids with boundary layer yields the least discrepancy against results obtained experimentally. The overall discrepancy obtained was 26.1%. This followed by the Hexahedral, Hybrid with no boundary layers and tetrahedral grids generation methods, with overall discrepancy of 35.6%, 40.0% and 41.2% respectively. For the grid generation method implemented in AVL SWIFT, it can be seen from Table 4.1 and Figure 4.14 that the Polyhedral Grids with boundary layers yields the least discrepancy against experimental results with 21.9% overall discrepancy. The Polyhedral Grids without boundary layers yields an overall discrepancy of 38.7%.

Figure 4.15: Comparison between Hexahedral, Hybrid and Tetrahedral Grid Generation Method in FLUENT at 0° Yaw
Figure 4.16: Grid Independency Test Results for the Slanted Edge Model in AVL at 0° Yaw with Standard Wall Function (br – bottom row at 13, 000, 34, 000, 107, 000, 199, 000, 820, 000, 1.5 million and experimental at 60 km/h)

From the analysis of all the grid generation methods employed for the sharp edge model at 0° yaw, it can be concluded that the most feasible method chosen is the Polyhedral Grids with boundary layers, which was generated using AVL SWIFT. The polyhedral grids were chosen because it was easy to generate having the least overall discrepancy. The amount of grids generated was similar to methods used in FLUENT. This grid generation method will be used in AVL SWIFT to assess the performance of various near wall and turbulence models. The hybrid grids with boundary layers will be used to assess the performance of various near wall and turbulence models in FLUENT.
Figure 4.17: Comparison between Hybrid and Polyhedral Grids at High and Low Reynolds Number Conditions
Table 4.1: Percentage in Discrepancy of Results between GAMBIT and AVL Fame Hybrid Grid Generation Methods against Experimental Results

<table>
<thead>
<tr>
<th>Grid Generation Strategy for Slanted Edge Model at 0° Yaw</th>
<th>Percentage of Discrepancy from Experimental Value (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Overall Discrepancy</td>
</tr>
<tr>
<td>High Reynolds $k$-$e$ Turbulence Model with Hexahedral Grids in FLUENT</td>
<td>35.6%</td>
</tr>
<tr>
<td>High Reynolds $k$-$e$ Turbulence Model with Tetrahedral Grids in FLUENT</td>
<td>41.2%</td>
</tr>
<tr>
<td>High Reynolds $k$-$e$ Turbulence Model with Hybrid Grids in FLUENT</td>
<td>40.0%</td>
</tr>
<tr>
<td>Low Reynolds $k$-$e$ Turbulence Model with Hybrid Grids in FLUENT</td>
<td>26.1%</td>
</tr>
<tr>
<td>High Reynolds $k$-$e$ Turbulence Model with Polyhedral Grids in AVL SWIFT</td>
<td>38.7%</td>
</tr>
<tr>
<td>Low Reynolds $k$-$e$ Turbulence Model with Polyhedral Grids in AVL SWIFT</td>
<td>21.9%</td>
</tr>
</tbody>
</table>
4.2.4 Near Wall Model and Turbulence Model Feasibility Study

Feasibility study using a combination of near wall models and turbulence models were conducted in this project to assess the suitable near wall model and turbulence model for the CFD model for this project.

4.2.4.1 Circular Models at 0º Yaw - Near Wall Model and Turbulence Model Feasibility Study

The results obtained from the time averaged RANS CFD simulation was presented as graph of the coefficient of surface mean pressure ($C_p$) versus the non-dimensionalised distance of the 16 total pressure-tapping points. For the SE, Semi and LE model, the datum point was located at 470, 490 and 530 mm from the centre of windshield respectively. First point of measurement ($x/L = 0.02$) was 10 mm from the initial datum point. The total distance for all 16 pressure-tapping points was 480 mm.

From Figures 4.18, 4.19 and 4.20, the bottom and top row monitoring locations behind the A-pillar region was plotted on the right and left of the graphs respectively. Comparisons were conducted between CFD and experimental data of Alam (2000) obtained at inlet free-stream velocity of 60, 100 and 140 km/h.

For the circular models at 0º yaw, simulation was conducted using the AVL SWIFT CFD package with standard $k – \varepsilon$ turbulence model and near wall model of Chieng and Launder (1980). For the purpose of simplicity, the standard $k – \varepsilon$ turbulence model and the NWM of Chieng and Launder (1980) will be referred to as SKE and 2LWF respectively. The 2LWF was chosen ahead of the standard wall function of Launder and Spalding (1974) in an effort to better predict and accommodate the changes of turbulent kinetic energy production in the viscous sub-layer, due to pressure gradient effect, thereby reducing the influence of the near wall cell size.
From Figures 4.18, 4.19 and 4.20, overall results obtained for the circular models at 0° yaw showed good correlations with the experimental results of Alam (2000). Minimum discrepancies were observed in the bottom row monitoring locations. Mean error due to the deviation of results obtained from the CFD simulation for the SE, Semi and LE model were 8.2%, 8.5% and 11.8% respectively. This is within the allowable percentage mean error deviation of 15% - 20% normally obtained from the results of CFD modelling, Swinburne (2000). A significant deviation of error was obtained from the CFD results in the LE model (Figure 4.18). This was contributed from the fact that results obtained by Alam (2000) at inlet free stream velocity of 60 km/h deviated significantly from the mean results obtained from inlet free stream velocity of 100 and 140 km/h respectively. Therefore, since the discrepancies values are within allowable range, the SKE and 2LWF turbulence and wall model will be used as the final CFD model for circular model at 0° yaw.
Figure 4.19: Semi Model, 0° Yaw

Figure 4.20: LE Model, 0° Yaw
4.2.4.2 Circular Models at 5°, 10° and 15° Yaw – Near Wall Model and Turbulence Model Feasibility Study

In this section, comparison between turbulence model of SKE, HTM2, $k – \omega$ and RSM was conducted using their respective accompanying near wall model. For the SKE turbulence model, the low Reynolds number model of Lam and Bremhorst (1981) will be used. For the purpose of simplicity, the low Reynolds number model of Lam and Bremhorst (1981) will be referred to as the LB model. The Wilcox (1998) $k – \omega$ turbulence model didn’t require any near wall model to model near wall effect since the turbulence model was developed to handle modelling of near wall effect. The RSM will be modelled using the Wall Elliptic Blend model of Manceau and Hanjalic (2002). For the purpose of simplicity, Wall Elliptic Blend model of Manceau and Hanjalic (2002) will be referred to as the WEB model. Finally, the HTM2 turbulence model will be modelled using the 2LWF near wall model. Although the 2LWF near wall model does not require mesh refinement all throughout until the viscous sub-layer, the attractiveness of this turbulence model in offering the combination of the SKE and RSM model obtained the inclusion in this investigation. For simplicity, the turbulence models and their respective near wall model will be referred to as the SKE-LB model, the $k – \omega$ model, the HTM2-2LWF model and the RSM-WEB model. The turbulence models and their respective near wall model assessment were conducted at yaw angles of 5°, 10° and 15° and at 140 km/h. Comparisons were made against the experimental data of Alam (2000).
Figure 4.21: SE Model, 5° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.22: SE Model, 10° Yaw, Bottom Row, Turbulence Model Comparison
Figure 4.21 showed results obtained for the SE model at 5° yaw. Figures 4.22 correspond to SE model at 10° yaw while figures 4.23 correspond to SE model at 15° yaw respectively. From the Figures at 5° yaw, the SKE-LB model, HTM2-2LWF model, \( k - \omega \) model and RSM-WEB model produced an overall mean percentage error deviation of 20.5%, 20.0%, 18.9% and 16.8% from the experimental results respectively. From the comparisons, it can be seen that the RSM-WEB model produced the best result in predicting the flow phenomena behind the A-pillar region for the SE model at 5° yaw. At 10° yaw, the SKE-LB model, HTM2-2LWF model, \( k - \omega \) model and the RSM-WEB model produced an overall mean percentage error deviation of 26.2%, 19.2%, 20.7% and 18.8% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the RSM-WEB model produced the best result in predicting the flow phenomena behind the A-pillar region for the SE model at 10° yaw. CFD modelling results obtained at 15° yaw was similar as the ones obtained at 10° yaw. The SKE-LB model, HTM2-2LWF model, \( k - \omega \) model and the RSM-WEB model produced an overall mean percentage error deviation of 32.1%, 26.3%, 28.9% and 20.3% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the RSM-WEB model produced the best
result in predicting the flow phenomena behind the A-pillar region for the SE model at 15° yaw.

Figure 4.24: Semi Model, 5° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.25: Semi Model, 10° Yaw, Bottom Row, Turbulence Model Comparison
For the Semi model, results for the turbulence model and near wall model assessment can be seen in Figures 4.24 for results at 5° yaw. Figures 4.25 correspond to Semi model at 10° yaw while Figures 4.26 correspond to the Semi model at 15° yaw respectively. The overall results obtained at 5° yaw showed the SKE-LB model, the HTM2-2LWF model, the $k-\omega$ model and the RSM-WEB model produced an overall deviation of 25.1%, 17.3%, 14.5% and 14.5% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the $k-\omega$ model and the RSM-WEB model produced the best result in predicting the flow phenomena behind the A-pillar region for the Semi model at 5° yaw. At 10° yaw the overall results showed that the SKE-LB model, the HTM2-2LWF model, the $k-\omega$ model and the RSM-WEB model produced an overall deviation of 25.3%, 18.2%, 15.9% and 15.8% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the RSM-WEB model produced the best result in predicting the flow phenomena behind the A-pillar region for the Semi model at 15° yaw. At 15° yaw the overall results showed that the SKE-LB model, the HTM2-2LWF model, the $k-\omega$ model and the RSM-WEB model produced an overall deviation of 37.1%, 28.1%, 22.1% and 18.1% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the RSM-WEB model produced the best result in
predicting the flow phenomena behind the A-pillar region for the Semi model at 15° yaw.

Figure 4.27: LE Model, 5° Yaw, Bottom Row, Turbulence Model Comparison

Figure 4.28: LE Model, 10° Yaw, Bottom Row, Turbulence Model Comparison
Figure 4.29: LE Model, 15° Yaw, Bottom Row, Turbulence Model Comparison

For the LE model, Figures 4.27 showed results obtained for the LE model at 5° yaw. Figures 4.28 correspond to LE model at 10° yaw while figures 4.29 correspond to LE model at 15° yaw respectively. The overall results obtained at 5° yaw showed that the SKE-LB model, the HTM2-2LWF model, the \( k-\omega \) model and the RSM-WEB model produced an overall deviation of 23.7%, 17.2%, 17.7% and 14.1% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the RSM-WEB model produced the best result in predicting the flow phenomena behind the A-pillar region for the LE model at 5° yaw. At 10° yaw the overall results showed that the SKE-LB model, the HTM2-2LWF model, the \( k-\omega \) model and the RSM-WEB model produced an overall deviation of 22.9%, 18.5%, 17.1% and 18.5% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the \( k-\omega \) model produced the best result in predicting the flow phenomena behind the A-pillar region for the LE model at 10° yaw. At 15° yaw the overall results showed that the SKE-LB model, the HTM2-2LWF model, the \( k-\omega \) model and the RSM-WEB model produced an overall deviation of 31.3%, 26.7%, 25.7% and 24.5% from the experimental results respectively. From the turbulence model comparisons, it can be seen that the RSM-WEB model produced the best result in
predicting the flow phenomena behind the A-pillar region for the LE model at 15° yaw.

Figure 4.30: SE Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.31: SE Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model
From the comparative investigation conducted using the SE model, the RSM-WEB model produced results with the minimum percentage error deviation. At 5°, 10° and 15° yaw, the percentage error of deviation are 16.8%, 18.8% and 20.3% (Figure 4.30, 4.31, 4.32). For the Semi model, the RSM-WEB model produced results with the minimum percentage error deviation. At 5°, 10° and 15° yaw, the percentage error of deviation are 14.5%, 15.8% and 18.1% (Figure 4.33, 4.34, 4.35). Finally, for the LE model, the RSM-WEB model produced results with the minimum percentage error deviation. At 5°, 10° and 15° yaw, the percentage error of deviation are 14.1%, 18.5% and 24.5% (Figure 4.36, 4.37, 4.38). It can be concluded that the RSM-WEB model is the most feasible near wall and turbulence model combination for the circular models at yaw angles and will be used as the CFD model for further analysis later in this chapter. The percentage error of deviation for the RSM-WEB is within allowable values recommended, Swinburne (2000).
Figure 4.33: Semi Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.34: Semi Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model
From the study, it can be seen that the RSM-WEB model obtained improvement over the eddy viscosity models in modelling the circular models at various yaw angles due to better prediction of complex three dimensional flow that are subjected to curved surfaces, swirl, rotation, rapid variation of strain rate and boundary layer separation, FLUENT (2003), Wilcox (2002). These are implemented by modelling Reynolds stresses via transport equations, by passing the traditional method of the Eddy Viscosity method.
Figure 4.37: LE Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.38: LE Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model
4.2.4.3 Sharp Edge Models at 0°, 5°, 10° and 15° Yaw – Near Wall Model and Turbulence Model Feasibility Study

The sharp edge models investigated in this study are the SL and RE models at 0°, 5°, 10° and 15° yaw. Comparisons were first made against the experimental data of Alam (2000) on the performance of the SL model at 0° yaw using various $k$ – $\varepsilon$ turbulence models and near wall models and can be seen in Figures 4.39 and 4.40. The $k$ – $\varepsilon$ model used was of the SKE-LB model, RNG and Realizable $k$ – $\varepsilon$ models using the one equation near wall model of Wolfstein (1969). Comparisons were conducted for the model at inlet free-stream velocity of 140 km/h. From the figures, the Realizable model produced the least of error deviation at 55.6%, as compared to 61.0% and 69.6% for the RNG and SKE-LB models respectively.

Taking the results obtained from the Realizable $k$ – $\varepsilon$ and comparing with turbulence models of $k$ – $\omega$, HTM2-2LWF and RSM-WEB showed that the RSM-WEB produced results with the least error deviation at 29.8%, followed by the $k$ – $\omega$ model at 35.8% and the AVL HTM2 at 54.1% (Figures 4.41 and 4.42). Figures 4.43 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the SL model at 5° yaw. The under prediction of the relative $C_p$ distribution produced on the combined bottom and top row monitoring location yielded an average of 27.2%. Figures 4.44 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the SL model at 10° yaw. The under prediction of the relative $C_p$ distribution produced on the combined bottom and top row monitoring location yielded an average of 25.3%. Figures 4.45 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the SL model at 15° yaw. The under prediction of the relative $C_p$ distribution produced on the combined bottom and top row monitoring location yielded an average of 21.9%. The performance of the turbulence models are consistent as what was reviewed and discussed in section 3.6.
Figures 4.46 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the RE model at $0°$ yaw. The predicted error deviation of the relative $C_p$ distribution produced on the combined bottom and top row monitoring location yielded an average of 11.9%. Figures 4.47 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the RE model at $5°$ yaw. The predicted error of deviation of the relative $C_p$ distribution produced on the combined bottom and top
row monitoring location yielded an average of 15.2%. Figures 4.48 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the RE model at 10° yaw. The predicted error deviation of the relative $C_p$ distribution produced on the combined bottom and top row monitoring location yielded an average of 21.1%. Figures 4.49 showed the $C_p$ mean distribution behind the A-pillar region obtained from the RSM-WEB turbulence model for the RE model at 15° yaw. The predicted error deviation of the relative $C_p$ distribution produced on the combined bottom and top row monitoring location yielded an average of 19.5%.

Figure 4.41: SL Model, 0° Yaw, Bottom Row, Turbulence Model Comparison
From the comparative investigation conducted using the SL model, the RSM-WEB model produced results with the minimum percentage error deviation. At 0°, 5°, 10° and 15° yaw, the percentage error of deviation are 29.8%, 27.2%, 25.3% and 21.9%. For the RE model, the RSM-WEB model produced results with the minimum percentage error deviation. At 0°, 5°, 10° and 15° yaw, the percentage error of deviation are 11.9%, 15.2%, 21.1% and 19.5%. It can be concluded that the RSM-WEB model is the most feasible near wall and turbulence model combination for the sharp edge models and will be used as the CFD model for further analysis later in this chapter. The percentage error of deviation for the RSM-WEB is within allowable values recommended for the RE model and slightly above allowable values recommended for the SL model, Swinburne (2000).
Figure 4.43: Slanted Edge Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.44: Slanted Edge Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.45: Slanted Edge Model, 15° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.46: Rectangular Model, 0° Yaw, Bottom Row, RSM-WEB Turbulence Model
Figure 4.47: Rectangular Model, 5° Yaw, Bottom Row, RSM-WEB Turbulence Model

Figure 4.48: Rectangular Model, 10° Yaw, Bottom Row, RSM-WEB Turbulence Model
4.3 Circular Models at 0º Yaw – Results and Discussion

CFD results obtained from the SKE-2LWF model were presented as external airflow streamline in Figures 4.50 and 4.51. The flow behaviour was in agreement with Barnard (1996). It can be seen that the airflow movement around the vehicle starts from the junction of the A-pillar base. The circular apex of the A-pillar causes separation, which was similar to a trailing edge separation (Figures 1.2 and 1.3; page 5 and 7). This resulted in the formation of boundary layer close to the vehicle wall surface. The boundary layer thickness increase as the airflow progressed downstream. The formation of turbulent boundary layer originated in laminar form and the laminar effect causes airflow to slide over each other due to skin friction drag formed with the vehicle wall surface. This will cause the outer air layer moving faster than the inner one which will slow down the flow, causing boundary layer to gradually become thicker. As the formation of turbulent boundary layer progressed, the flow becomes streamlined, following the contours of the vehicle body. The small scale turbulent eddies are causing rapid mixing. However, very close to the surface within a turbulent boundary layer flow, a thin sub layer of laminar flow still exists (Figure 4.52, page 167). This result in a gradual increase in relative pressure as the airflow moves downstream, which will have a direct impact on the process of turbulent mixing. It will allow energy
transfer to take place from the fast moving eddies to the slower moving eddies in the turbulent boundary layer resulting in turbulent boundary layer with low turbulent activity.

Figure 4.50: SE Model, 0° Yaw, External Surface Streamline, Front View

Figure 4.51: SE Model, 0° Yaw, External Surface Streamline, Top View
The turbulent boundary layer exists in a quasi two-dimensional form. This can be seen from the velocity vector generated in Figure 4.53. Measurement of the cross section indicate a horizontal and vertical separation of 115-mm and 102-mm. Comparison against Semi and LE model showed no significant difference in either the shape and form of the separation. Correlation between surface streamline flow of the model and the streamline visualization using wool tuffs by Alam (2000) showed that the turbulent boundary layer close to the surface is attached and quasi two-dimensional (Figures 4.54 and 4.55). The quasi two-dimensional separation that was observed was in agreement with Hucho (1998). It can be seen from the Figures that the quasi two-dimensional type flow separates at the A-pillar apex running and almost perpendicular to the local direction of flow. The separation causes vortices to roll up with their axes almost parallel to the separation line forming a circular and elongated quasi two-dimensional separation downstream to the flow. Turbulent mixing dissipates most of their kinetic energy making their development as continuing free trailing vortices, often weak and even untraceable. Often, two-dimensional separation forms a continuing trailing vortex with three-dimensional separation forms a cone shaped helical vortex. Correlation in Figure 4.18 against Alam (2000) showed that Cp distribution at various distance along the A-pillar/side wall was Reynolds number independent and that most importantly, flow separation exists during the start of the flow, growing weaker as it moves downstream.
Furthermore from Figure 4.51, the quasi two-dimensional turbulent boundary layer separation forms a region defined as ‘separation bubble’, Hucho (1998).
Separation bubble is defined as the region between flow separation and reattachment caused by adverse pressure gradient from the rapid change of pressure at the A-pillar apex. The separation bubble region will form region of vortex circulation termed the ‘dead water’ zone. According to Watanabe et al. (1978), the vortex development forms region of low relative pressure and that the core is located at the vortex core of the separation. This is in agreement with Figure 4.18 where region of low relative pressure was observed occurring at the vortex core located at the apex of the A-pillar. Observation from the Semi and LE models in Figures 4.56 and 4.57 showed similar magnitude and pattern of distribution for turbulent velocity as compared with the SE model (Figure 4.52). However, observation seen from Figures 4.18 to 4.20 showed a decrease in Cp as the windshield radii of the models increases. This is in agreement with Callister et al. (1998). An alternative way of reducing A-pillar vortex size and intensity was to increase the surface area of the models by increasing the windshield/A-pillar radius.

Figure 4.54: SE Model, 0° Yaw, Surface Streamline
Figure 4.55: SE Model at 0° Yaw, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)

Figure 4.56: Semi Model, 0° Yaw, Top View, Turbulent Velocity
4.4 Circular Models at 5° yaw – Results and Discussion

The surface flow streamline for the SE, Semi and LE model at 5° yaw can be seen in Figures 4.58 to 4.61. It can be seen from these figures that the effect of yaw angle is creating bigger separation regions around and behind the A-pillar in the leeward region as oppose to the windward region of the flow. This observation is in agreement with Alam et al. (1998), Bearman et al. (1989) and Haruna et al. (1990). The presence of yaw angle resulted in a larger separation on the leeward side due to an increased in turbulence intensity (Figures 4.62 and 4.63). Figures 4.62 and 4.63 of the velocity vector and turbulent velocity show a larger separation in the leeward region at 5° yaw. Measurement of the cross section indicate a horizontal and vertical separation of 150-mm and 370-mm. Observation against Semi and LE model showed no significant difference in either the shape and form of the separation. However, comparing Cp coefficient between the SE, Semi and LE model do show a decrease in negative relative pressure magnitude, which translate in a decline in turbulence intensity as the windshield radii increases (Figures 4.30, 4.33 and 4.36, page 152 to 155).
First Separation Region (Trailing Edge Vortex)

Second Separation Region (A-pillar Vortex)

Third Separation Region (Roof Junction)

Figure 4.58: SE Model, 5° Yaw, Leeward Region, External Flow Streamline

A-pillar Turbulent Boundary Layer Separation

Figure 4.59: SE Model, 5° Yaw, Windward Region, External Flow Streamline
It was observed from the surface flow streamline for each model that the source of flow separation in the leeward region originated from the corner of the windshield/A-pillar junction, the A-pillar apex and from the roof junction of the model. This observation was in agreement with Hanaoka et al. (1993), Ahmed (1998) and Zhu et al. (1993, 1994) in which it was noted that the regions of separated flow on a vehicle occur on the A-pillar base, together with the front side window and roof junction behind the A-pillar.

Figure 4.60: Semi Model, 5° Yaw, Front View, External Flow Streamline

It can also be seen in Figures 4.58 to 4.61, that in the windward side of the flow, the airflow originated from impingement on the model windshield, re-directing the flow to the windward side of the model. The induced air flow creates an attached turbulent boundary layer region as it moves around the curvature of the windshield in the A-pillar region downstream to the flow. The airflow flowing around the A-pillar region was observed spilling over the windward roof junction as attached flow.

The flow separation occurring in the leeward region was a combination of two-dimensional and three-dimensional, while the flow separation occurring in the windward region was mainly two-dimensional. It was observed from Figures 4.58 to 4.61 in the
leeward region that the separation region originated from the A-pillar base junction was quasi two dimensional. This was in agreement with Hucho (1998) in which he stated that the separation causes vortices to roll up with their axes almost parallel to the separation line. Turbulent mixing dissipates most of their kinetic energy making their development as continuing free trailing vortices.

The separation from the A-pillar apex in the leeward region was observed as three-dimensional, a mixture of separated and attached flow. The mixture of separation and attached flow behind the A-pillar region was attributed to the yaw angle of the vehicle. This was in agreement with Barnard (1996). Strong sideways or cross-stream components on the surface of a vehicle complicate the formation and behaviour of the boundary layer. Cross-stream components are more inclined to cause early transition of the turbulent boundary layer. Cross-stream flows can also keep the boundary layer attached by reducing high-pressure flow, making the pressure gradient less adverse. The separated region exhibit rotational flow while the attached region remains irrotational, Haruna et al (1990). The three dimensional separation normally occurs due to the effect of the

Figure 4.61: LE Model, 5° Yaw, Front View, Surface Streamline
inclination angle of the A-pillar resulting in a formation of a quasi circular/cone shaped helical vortex, Hucho (1998). The axes of these vortices run essentially in the stream wise direction. The vortex is very rich in kinetic energy. The containment of kinetic energy is determined by the A-pillar angle inclination. In addition, observation from Figures 4.58 to 4.61 indicates that the complex turbulence behaviour behind the A-pillar apex causes the vortex formed to stretch and breakdown experiencing a decrease in magnitude intensity in the direction normal to wall surface of the model and downstream to the flow. This is in line with the findings of Laufer (1974) and Hussain (1983, 1986).

Figure 4.62: SE Model, 5° Yaw, Front View, Velocity Vector

The flow separation from the trailing edge roof junction was observed mixing with vortex separation from the A-pillar, resulting in a mix of two-dimensional and three-dimensional separations with similar flow mechanism as the flow separation observed from the A-pillar apex.
Inspection from Figure 4.64 showed that the anisotropic nature of the turbulent flow behind the A-pillar region. The Reynolds stresses component distribution revealed different distribution in the UU (horizontal), VV (vertical) and WW (downstream) component, which disproves the usage of Boussinesq assumption in the eddy viscosity models in assuming isotropic turbulence, Wilcox (2002). The distribution of the Reynolds stresses components along the surface of the wall revealed the stream wise UU Reynolds stress component is around 52.3% of the total turbulent kinetic energy distribution. This is within close agreement to 60% for turbulent boundary layer distribution, Imperial College (circa. 1980s). The Reynolds stresses component also showed that along the surface of the wall, the leeward region of the flow displayed a higher Reynolds stresses intensity. This was due to the yawing orientation of the model that produces a higher Reynolds stresses component due to the vortex generation in the leeward side of the flow as compared to the windward side. The complex three-dimensional nature of the flow also revealed that the VV Reynolds stress component having a higher intensity compared to the WW Reynolds stress component as what was observed by Nienaltowska (1993) and in the standard free shear and flat plate turbulent
boundary layer flow, Imperial College (circa. 1980s). The Reynolds stresses distributions on the windward region were 37.0% higher compared to the distribution in the leeward region.

Figure 4.64: SE Model, 5° Yaw, Reynolds Stresses Component, Leeward and Windward Region of the A-pillar
4.5 Circular Models at 10° and 15° Yaw – Results and Discussion

The surface flow streamline for the SE, Semi and LE model at 10° and 15° yaw can be seen in Figures 4.65 to 4.70. When comparing these figures to those at yaw angles of 0° and 5° respectively, it was observed that the effect of yaw angle above 10° underwent a sudden escalation in separation regions and turbulence intensity around and behind the A-pillar in the leeward and windward region of the flow respectively. This observation is in agreement with Bearman et al. (1989). At yaw angle of 10° the separation originated from the A-pillar base junction and the A-pillar apex are subjected to higher adverse pressure gradient. Stronger sideways or cross-stream flows are present causing a much earlier transition of the turbulent boundary layer. As a result, larger area of separation and recirculation generated due to the higher yaw angle airflow approach. Region of low pressure close to the wall surface was present due to activities occurring beneath the turbulence boundary layer. The airflow surrounding the A-pillar region will be highly rotational and the vortex generated is high in kinetic energy. The highly turbulent nature will cause the vortex transient behaviour to fluctuate in size due to the act of mixing and dissipation of turbulent eddies. The size fluctuation will cause the vortex to periodically stretch and breakdown due to fluctuation in magnitude intensity in the direction normal to wall surface of the model and downstream to the flow. This is in line with the findings of Laufer (1974) and Hussain (1983, 1986).
As with observations at 5° yaw, it was observed from the surface flow streamline for each model at 10° and 15° yaw that the source of flow separation in the leeward region originated from the corner of the A-pillar base junction, the A-pillar apex and from the
roof junction of the model. The vortex generated from the A-pillar base junction formed a trailing helical shaped vortex while vortex generated from the A-pillar apex formed a combination of quasi elongated circular and cone helical vortex. The vortex generated from the roof junction formed a trailing circular shape vortex, exhibiting rotation and circulation flowing in stream wise direction. It can be seen from Figures 4.65 to 4.70 that the direction of flow generated by all three vortex sources interferes with each other creating a chaotic turbulent structure.

![Figure 4.67: Semi Model, 10° Yaw, Front View, External Flow Streamline](image)

It can also be seen in Figures 4.65 to 4.70, in the windward side of the flow, the airflow impinges on the model windshield, re-directing the flow to the windward side of the model. The induced air flow creates an attached turbulent boundary layer region as it moves around the curvature of the windshield in the A-pillar region downstream to the flow. The airflow flowing around the A-pillar region was also observed spilling over the windward roof junction as attached flow. At 10° and 15° yaw, the over spilling at the roof junction is higher forming a leading edge vortex circulating cross wise almost halfway across the roof.
In addition, Figures 4.75 to 4.78 of the turbulence velocity and velocity vector slice also shows a larger separation in the leeward region at 10° and 15° yaw. Observation against the SE, Semi and LE models showed no significant difference in both the shape and the mechanism of the separation (Figure 4.65 to 4.70). However, similar with the observation at 5° yaw, comparing Cp coefficient between the SE, Semi and LE model do show a decrease in negative relative pressure magnitude, which translate in a decline in turbulence intensity as the windshield radii increases. Measurement of the separation cross section for the model at 10° indicate a horizontal and vertical separation of 214-mm and 404-mm. Measurement of the separation cross section for the model at 15° indicate a horizontal and vertical separation of 340-mm and 430-mm. Correlation between surface streamline flow of the model and the streamline visualization using wool tuffs by Alam (2000) showed that the turbulent boundary layer close to the surface is attached and quasi two-dimensional (Figures 4.71 to 4.74).
The axis of the vortex core with respect to the horizontal plane was observed to increase as the yaw angle increases (Figures 4.54; page 168, 4.71; page 182 and 4.73; page 183). Axis of the vortex core for model subjected to 0° yaw was observed to be at 5° from the
horizontal plane. At the leeward region when subjected to 15° yaw, the vortex core axis was measured at an angle of 10° from the horizontal plane. At the windward region when subjected to 15° yaw, the vortex core axis was measured at an angle 20° respectively from the horizontal plane.

![Figure 4.71: SE Model, 15° Yaw, Leeward View, Surface Flow Streamline](image1)

![Figure 4.72: SE Model at 15° Yaw in the Leeward Region, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)](image2)
Figure 4.73: SE Model, 15° Yaw, Windward View, Surface Flow Streamline

20° vortex core axis relative to the horizontal axis

Figure 4.74: SE Model at 15° Yaw in the Windward Region, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)

Similar Flow Pattern as CFD model
Figure 4.75: SE Model, 10° Yaw, Top View, Turbulent Velocity

Figure 4.76: SE Model, 10° Yaw, Front View, Velocity Vector
Inspection from Figures 4.79 and 4.80 reveal the distribution of Reynolds stresses component were different in the UU, VV and WW component behind the A-pillar region.
Inspection from Figure 4.79 showed Reynolds stress component distribution at 10° yaw. Comparison to the distribution at 5° yaw showed several differences, which further show the anisotropic nature of the turbulent flow behind the A-pillar region. In the leeward region, the UU and VV component showed an increase production of turbulent kinetic energy and dissipation rate close to the wall surface region within the separation region. In the windward region, the Reynolds stress in the UU component showed a significant increase in comparison to the other component, which reflect on the fact that the flow in the windward region was predominantly attached, and moving in the stream wise direction, downstream to the flow. The stream wise UU Reynolds stress component is around 50.3% of the total turbulent kinetic energy distribution. This is within close agreement to 60% for turbulent boundary layer distribution along a flat plate, Imperial College (circa. 1980s). The Reynolds stresses distributions on the windward region were 35.0% higher compared to the distribution in the leeward region.

![Reynolds Stress Distribution, SE Model, 10 Degree Yaw, RSM-WEBC Model](image)

Figure 4.79: SE Model, 10° Yaw, Reynolds Stresses Component, Leeward and Windward Region of the A-pillar

Inspection from Figures 4.80 showed Reynolds stress component distribution at 15° yaw. The stream wise UU Reynolds stress component is around 52.8% of the total turbulent...
kinetic energy distribution. This is within close agreement to 60% for turbulent boundary layer distribution along a flat plate, Imperial College (circa. 1980s). The Reynolds stresses distributions on the windward region were 37.0% higher compared to the distribution in the leeward region.

Figure 4.80: SE Model, 15º Yaw, Reynolds Stresses Component, Leeward and Windward Region of the A-pillar
4.6 RE Model at 0° Yaw – Results and Discussion

CFD results obtained for the RE model at 0° yaw from the RSM-WEB model was presented as external airflow streamline in Figures 4.81. The airflow movement around the vehicle starts from vehicle windshield and separates from the A-pillar base junction, A-pillar edge and the roof junction.

The roof junction separation was similar to a trailing edge separation observed in the circular models. Separation from the roof junction formed a quasi two-dimensional flow resulted from the generation of adverse pressure gradient due to the rapid change in pressure. The separation at the roof junction causes vortices to roll up with their axes almost parallel to the separation line forming a continuing and trailing two dimensional vortex downstream to the flow. Turbulent mixing dissipates most of their kinetic energy making their development often weak and even untraceable, Hucho (1998).
The A-pillar edge separation formed a quasi two-dimensional separation bubble region, Hucho (1998). The rapid change of pressure at the A-pillar edge causes formation of adverse pressure gradient between the region of flow separation and reattachment. This will also result in a dead water vortex circulation zone formation downstream to the flow. According to Watanabe et al. (1978), the separation bubble forms region of low relative pressure in which the core is located at the vortex core of the separation.

Figure 4.82: Rectangular Model, 0° Yaw, Side View, Surface Streamline

Figure 4.83: RE Model at 0° Yaw, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)
The A-pillar base junction separation formed three dimensional vortices that are very rich in kinetic energy which are determined by the A-pillar inclination angle, Hucho (1998). Figure 4.81 and 4.82 showed helical vortex originating from the A-pillar base junction and bonnet, propagating around 55° relative to the horizontal axis downstream to the flow, Murad (2004). Turbulent velocity contour slice in plan view in Figure 4.84 shows helical vortex formation A-pillar base junction propagating downstream to the flow. It can also be seen that the helical vortex propagation is surrounded by separation bubble formed from the A-pillar edge separation. The resultant formation of separation bubble is evident in green contour gradient and acts as a larger but less intense region of separation surrounding the helical vortex from the A-pillar base junction.

From Figure 4.84, measurement of the helical vortex cross section indicate a horizontal and vertical separation of 80-mm and 200-mm. Correlation between surface streamline flow of the model and the streamline visualization using wool tuffs by Alam (2000)
showed that the turbulent boundary layer close to the surface is attached and quasi two-dimensional (Figures 4.82 and 4.83).

Figure 4.85: Rectangular Model, 0° Yaw, Front View, UU Reynolds Stress Component

Figure 4.86: Rectangular Model, 0° Yaw, Front View, VV Reynolds Stress Component
Inspection from Figures 4.85 to 4.87 showed that the anisotropic nature of the turbulent flow behind the A-pillar region. The Reynolds stresses component distribution revealed different distribution in the UU, VV and WW component. Inspection from Figures showed that the UU has the highest Reynolds stresses component distribution followed by the VV and WW component respectively. This is within close agreement to the circular models seen in previous sections and also with published literature (Imperial College, circa. 1980s).

### 4.7 SL Model at 0° Yaw – Results and Discussion

CFD results obtained for the SL model from the RSM-WEB model was presented as external airflow streamline in Figure 4.88. The airflow movement around the SL model starts from vehicle windshield experienced separation at the A-pillar region, in which components consists of the A-pillar base junction, A-pillar edge and the roof junction.
Airflow from the vehicle windshield experienced trailing separation at the A-pillar region. Due to the A-pillar slant angle, separation bubble takes shape in the form of three dimensional conical helical vortex, which was formed approximately 60° from the vertical axis, parallel to the A-pillar slant angle. The vortex formation was the result of rapid pressure change, forming region of adverse pressure gradient between the area of flow separation and reattachment. Rapid pressure change between layers of airflow behind the A-pillar edge formed a forced vortex phenomena, Roberson et al. (1997). Viscous slipping between adjacent layers of fluid molecules forces high air velocity flow on the outer layer of the vortex glide pass the inner layer vortex formation. The airflow velocity increases linearly from the vortex centre. Trailing separation from the A-pillar edge and the roof junction forces the vortex to form an elongated separation bubble, which was stretched downward along the vertical plane, Murad (2004) (Figure 4.92). Plan view of the turbulent velocity contour showed the vortex being stretched downstream to the flow, Figure 4.91 (page 196). From Figure 4.91, low relative pressure was measured within the separation bubble region, in which where the vortex core was located. This can be seen in Figures 4.41 and 4.42 and was consistent with the findings of Watanabe et al. (1978).

Figure 4.88: Slanted Edge Model, 0° Yaw, Front View, External Flow Streamline
Measurements of the helical vortex cross section indicate a horizontal and vertical separation of 113-mm and 274-mm. The vortex generated from the A-pillar base junction, A-pillar apex and the roof junction combined to form a three-dimensional helical vertical elongated cone shaped vortex. The combination of these three vortex source creates a highly incensed turbulent structure propagating downstream to the flow.

Figure 4.89 showed that the helical vortex originating from the A-pillar base junction and bonnet, propagates around 30° relative to the horizontal axis downstream to the flow, Murad (2004). Correlation between surface streamline flow of the model and the streamline visualization using wool tuffs by Alam (2000) showed that the turbulent boundary layer close to the surface follows the same flow pattern as the separation vortex, as shown in Figures 4.89 and 4.90. Region of vortex reattachment can be seen halfway between the A-pillar edge and the A-pillar base junction. Correlation in Figures 4.40 and 4.41 (page 158 and 159) against Alam (2000) showed that Cp distribution at various distance along the A-pillar/side wall was Reynolds number independent and that most importantly, flow separation exists during the start of the flow, growing weaker as it moves downstream.
Figure 4.90: SL Model at 0° Yaw, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)

Figure 4.91: Slanted Edge Model, 0° Yaw, Turbulence Velocity Contour Distribution
Inspection from Figures 4.93 to 4.95 showed that the anisotropic nature of the turbulent flow behind the A-pillar region. The Reynolds stresses component distribution revealed different distribution in the UU, VV and WW component. Inspection from Figures showed that the VV (vertical) has the highest Reynolds stresses component distribution followed by the UU (horizontal) and WW (downstream) component respectively.
Figure 4.93: Slanted Edge Model, 0° Yaw, Front View, UU Reynolds Stress Component

Figure 4.94: Slanted Edge Model, 0° Yaw, Front View, VV Reynolds Stress Component
External surface streamline airflow observed for the Rectangular model at 5° yaw shows flow separation both at the leeward and windward region of the A-pillar (Figure 4.96). The 5° yaw causes the airflow to impinge slightly off centre on the windshield surface. Trailing edge separation was observed originating from the junction of the A-pillar base, the A-pillar edge and the roof junction. Quasi two-dimensional flow separation was observed from the A-pillar base junction while a combination of two-dimensional and three-dimensional flow separation was observed coming from the A-pillar apex and roof junction. The perpendicular angle between the bonnet and windshield, also the windshield and the roof causes quasi two-dimensional type flow to separate on the edge and run perpendicular to the local direction of flow and interfering with each other creating a chaotic turbulent structure. The separation causes vortices to roll up with their axes almost parallel to the separation line. This was particularly evident at the A-pillar edge and roof junction. Coupled with the yaw angle, the separation causes turbulent
mixing, which translate to a larger separation bubble and enhanced turbulent intensity (Figure 4.99). Kinetic energy generated from the separation then dissipates downstream as continuing free trailing vortices, Hucho (1998). The overall separation bubble, which was a result of the chaotic turbulent structure, takes the form of a quasi circular elongated separation vortex stretched downstream to the flow. The physical shape of the vortex for the RE model at yaw angles are similar to those of the circular models. However, the turbulent intensity are higher in the RE model due to the influence of the sharp A-pillar edge as compared to the streamlines curved A-pillar in the circular models. Measurements of the separation bubble in the leeward region indicate a cross and vertical separation of 162-mm and 487-mm. Measurement in the windward region indicate a cross and vertical separation of 44-mm and 267-mm.

Figure 4.96: Rectangular Model, 5° Yaw, Frontal External Streamline Airflow
In the windward region of the A-pillar, the effect of the yaw angle, together with the vertical windshield plane (Popat, 1991 and Hucho, 1998) causes the flow separation bubble to be smaller but higher in turbulence intensity compared to the separation occurring in the leeward region (Figures 4.96 and 4.47). The effect of the yaw angle and the vertical windshield plane in the windward region forces the airflow to separate at the edge of the A-pillar, having a smaller region of separation, recirculation and reattachment thus generating high concentration of quasi two-dimensional turbulent flow mixing, Murad (2004).
As the yaw angle increases to 10° and 15° yaw, the yaw angle causes impingement of the airflow on the windshield to move closer to the windward region of the A-pillar. The
increase in yaw angles causes the separation bubble in the leeward and windward region of the A-pillar to become bigger. At 10° yaw, measurement of the separation bubble in the leeward region indicate a cross and vertical separation of 302-mm and 554-mm. Measurement in the windward region indicate a cross and vertical separation of 53-mm and 264-mm. At 15° yaw, measurement of the separation bubble in the leeward region indicate a cross and vertical separation of 450-mm and 590-mm. Measurement in the windward region indicate a cross and vertical separation of 25-mm and 326-mm.

Figure 4.100: Rectangular Model, 15° Yaw, Leeward Surface Streamline Airflow
At 10° and 15° yaw, the mechanism of flow separation and vortex generation together with the overall physical shape of the separation bubble is similar to that at 5° yaw. However, several differences were observed in the behaviour of the flow separation due to the increased yaw angle. As the yaw angle increase, the vortex core axis decreases when measured with respect to the A-pillar horizontal axis (Figures 4.100 and 4.82). The vortex core was measured at 55° at 0° yaw and at 38° at 15° yaw.

Correlation between surface streamline flow of the model and the streamline visualization using wool tuffs by Alam (2000) showed that at 15° yaw the turbulent boundary layer close to the surface follows the same flow pattern as the separation vortex, Figures 4.100 to 4.103. The pressure measurements made on the vehicle surface (Figures 4.47, 4.48 and 4.49) can be associated to describe the intensity and behaviour of the flow separation behind the A-pillar region, Watanabe et al. (1978), Bearman et al. (1989) and Alam (2000). From the pressure measurements made on the vehicle surface
behind the A-pillar region, it can be seen that as the yaw angle increases, the vortex strength on the leeward region decrease with an increase in separation bubble region. Simpson (1987, 1989) explained that the large-scale turbulent eddy motions produced in the vortex flow separation do not contribute much in influencing the increase of turbulent shear stresses. However, in the windward region, the flow separation behaviour is independent of the yaw angle increase. The vortex peaked at 10° yaw before experiencing a decline at 15° yaw. The separation bubble region became smaller as the vehicle yaw angle increases.

Figure 4.102: Rectangular Model, 15° Yaw, Windward Surface Streamline Airflow
4.9 Slanted Model at 5°, 10° and 15° Yaw – Results and Discussion

CFD results obtained for the SL model from the RSM-WEB model at 5°, 10° and 15° yaw was presented as external flow streamline in Figures 4.104, 4.105 and 4.106 respectively. When the SL model was subjected to these various yaw angles, it was observed that at the leeward region of flow, the intensity and size of the helical conical turbulence vortex formed behind the A-pillar region increases, Murad (2004). In the windward region of the flow, the intensity and size of the turbulent vortex is less than what was observed in the leeward region. This was the opposite as to what was observed from the RE model in the previous section, when exposed to airflow at various yaw angles. In the leeward region at 5° yaw, the size of the vortex separation in the horizontal and vertical direction was measured at 217-mm and 358-mm respectively (Figure 4.104). At 10° yaw, the size of the vortex separation in the horizontal and vertical direction was measured at 310-mm and 320-mm respectively (Figure 4.105). Finally, at 15° yaw, the size of the vortex separation
in the horizontal and vertical was measured at 440-mm and 280-mm respectively (Figure 4.106). Size of the vortex separation in the windward region at 5° yaw was measured at 31-mm and 348-mm in the horizontal and vertical direction respectively. At 10° yaw, the vortex separation was measured at 19-mm and 330-mm in the horizontal and vertical direction respectively. Finally, at the 15° yaw, the vortex separation was measured at 0-mm and 294-mm respectively. Measurement and observation made from the figures indicate that the vortex core drifts away from the model surface in the leeward region as the yaw angle increases. In the windward region, the vortex core shifted more towards the surface as the yaw angle increases (Figure 4.104).

Airflow impinging the SL model windshield surface when subjected to yaw angle experienced three-dimensional trailing separation at the A-pillar region (Figure 4.105). Due to the yaw angle, airflow impinges on the windshield surface offset from centre and due to this; it will have strong sideways or cross-stream components of velocity on the windshield surface as it travels to the leeward region, causing early boundary layer transition, Barnard (1996). Early boundary layer transition coupled with trailing
separation at the slanting A-pillar edge causes turbulent intensity and size to increase as the yaw angle increase. The axis of the three dimensional conical helical vortex core was observed to increase as the yaw angle increases (Figures 4.108 and 4.89). At the leeward region when subjected to 15° yaw, the vortex core axis was measured at an angle of 45° from the horizontal plane. When model subjected to 0° yaw, it was observed to be at 30° from the horizontal plane.

Figure 4.105: Slanted Model, 10° Yaw, Frontal External Streamline Airflow
It can be seen from the external flow streamline from Figures 4.104 to 4.106 that at the leeward region, flow from the A-pillar base junction, the A-pillar slant edge and the roof
junction combine to form the three dimensional conical helical vortex, making the turbulent intensity stronger as the yaw angle increases. Changing these parameters effects the intensity, size and shape of the vortex generated behind the A-pillar region, Popat (1991). Observation of various shapes of windshield radii and slant angle models simulated in this project showed that high intensity three dimensional vortex was generated when airflow separation originated from various parts of the surface edge combine together effectively. This is possible when airflow path from various separation surface are travelling in the same direction, before and after separation in order to encourage a consistent vortex core axis all the way downstream to the flow. The geometry and radii of the windshield determine the intensity of the A-pillar vortex based on the convergence of these localised separations. The resultant high intensity A-pillar vortex produces high velocity region within the vortex, which translate to a region of low negative pressure causing a vacuum like effect, Roberson et al. (1997). In the case of the SL model, combination of sharp edges with various geometrical angles at three different locations (A-pillar base, A-pillar apex and Roof junction) provided a good combination in generating the three-dimensional helical conical vortex due to the trailing separation generated behind the A-pillar region. As a result, it can be seen in Figures 4.43, 4.44 and 4.45 that for the SL model at yaw angles, the Cp value, which represents the strength of pressure inside the vortex formation have a pronounced peak and its magnitude is higher compared to the other models simulated. Correlation between surface streamline flow of the model and the streamline visualization using wool tufts by Alam (2000) showed that at 15° yaw the turbulent boundary layer close to the surface follows the same flow pattern as the separation vortex, Figures 4.108 to 4.111.
Figure 4.108: Slanted Model, 15° Yaw, Leeward Surface Streamline Airflow

45° vortex core axis relative to the horizontal axis

Figure 4.109: SL Model at 15° Yaw in the Leeward Region, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)
Figure 4.110: Slanted Model, 15° Yaw, Windward Surface Streamline Airflow

Figure 4.111: SL Model at 15° Yaw in the Windward Region, Surface Streamline Visualisation using Wool Tuffs (after Alam, 2000)
4.10 General Discussion

For the circular models (SE, Semi and LE) at 0°, the final turbulence and NWM chosen for the project was the standard $k - \varepsilon$ turbulence model and the NWM of Chieng and Launder (1980). For the circular models at 5°, 10° and 15° yaw angles, together with the sharp edge models at 0°, 5°, 10° and 15° yaw, the final turbulence and NWM model chosen for the project was the RSM with the WEB NWM of Manceau and Hanjalic (2002). The RSM and WEB NWM model performs better in predicting three-dimensional flow through direct modelling of the Reynolds stresses and the redistribution term as compared to the eddy viscosity models. The RSM and WEB NWM model held an average of 1.97% improvement over the eddy viscosity turbulence and it’s respective near wall models for the circular models. It held a 2.0% improvement for the sharp edge models.

Results and analysis from this chapter has shown that comparison of mean Cp values between the CFD and experimental results have shown that for the circular models (SE, Semi and LE), error of deviation for models at 0°, 5°, 10° and 15° yaw angle have all fallen within the recommended 20% margin (Table 4.2).

Table 4.2: Percentage Error Deviation of Models against Results of Alam (2000) at various Yaw Angles

<table>
<thead>
<tr>
<th>Models</th>
<th>Yaw Angle (° degrees)</th>
<th>0°</th>
<th>5°</th>
<th>10°</th>
<th>15°</th>
</tr>
</thead>
<tbody>
<tr>
<td>SE</td>
<td></td>
<td>8.2%</td>
<td>16.8%</td>
<td>18.8%</td>
<td>20.3%</td>
</tr>
<tr>
<td>Semi</td>
<td></td>
<td>8.5%</td>
<td>14.5%</td>
<td>15.8%</td>
<td>18.1%</td>
</tr>
<tr>
<td>LE</td>
<td></td>
<td>11.8%</td>
<td>14.1%</td>
<td>18.5%</td>
<td>24.5%</td>
</tr>
<tr>
<td>RE</td>
<td></td>
<td>11.9%</td>
<td>15.2%</td>
<td>21.1%</td>
<td>19.5%</td>
</tr>
<tr>
<td>SL</td>
<td></td>
<td>29.8%</td>
<td>27.2%</td>
<td>25.3%</td>
<td>21.9%</td>
</tr>
</tbody>
</table>
The best performing model was the SE model at 0° which yielded an under prediction of 8.2%. The worse performing model was the LE model at 15° yaw which yielded an under prediction of 24.5%. For sharp edge models (RE and SL) error of deviation for models at yaw angles of 0°, 5°, 10° and 15° yaw has also fallen within the recommended 20% margin. The best performing model was the RE model at 0° which yielded an under prediction of 11.9%. The worse performing model was the SL model at 0° yaw which yielded an under prediction of 29.8%. Under predictions obtained through the turbulence and near wall models occur within the vortex core area, in which the airflow were experiencing vortex separation and reattachment. Comparison of airflow on the vehicle surface between the CFD results and Alam (2000) showed that good correlation was obtained. Comparisons were made for airflow streamline on the vehicle surface between visuals obtained during the CFD post processing stage in the CFD model against wool tuffs visualisation technique implemented by Alam (2000), which showed good correlation with each other.

The size of the vortex measured for the various models at different yaw angles in the leeward region of the flow suggest that with the circular models (SE, Semi and LE) and the sharp edge models (RE and SL), the vortex size increases with yaw angles. This was in agreement with Bearman et al. (1989) and Haruna et al. (1990). Refer Table 4.2. Vortex measurement for the circular models at 40% scale shows that as the yaw angle increases from 0° to 15° yaw, the vortex experiences a size increment of 225 mm in the horizontal component and 328 mm in the vertical component. For the circular models, it can be seen that the vortex is larger in the vertical component (430 mm at 15° yaw). The vertical component also experiences a higher increase with an increase in yaw angles.

<table>
<thead>
<tr>
<th>Yaw Angles</th>
<th>0°</th>
<th>5°</th>
<th>10°</th>
<th>15°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal (mm)</td>
<td>115</td>
<td>150</td>
<td>214</td>
<td>340</td>
</tr>
<tr>
<td>Vertical (mm)</td>
<td>102</td>
<td>370</td>
<td>404</td>
<td>430</td>
</tr>
</tbody>
</table>
Vortex measurement for the RE model at 40% scale shows that as the yaw angle increases from 0° to 15° yaw, the vortex experiences a size increment of 370 mm in the horizontal component and 390 mm in the vertical component. For the RE model, it can be seen that the vortex is larger in the vertical component (590 mm at 15° yaw). The vertical component also experiences a higher increase with an increase in yaw angles. Refer Table 4.3.

Table 4.4: RE Model Vortex Size at 40% scale

<table>
<thead>
<tr>
<th>Yaw Angles</th>
<th>0°</th>
<th>5°</th>
<th>10°</th>
<th>15°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal (mm)</td>
<td>80</td>
<td>162</td>
<td>302</td>
<td>450</td>
</tr>
<tr>
<td>Vertical (mm)</td>
<td>200</td>
<td>487</td>
<td>554</td>
<td>590</td>
</tr>
</tbody>
</table>

Vortex measurement for the SL model at 40% scale shows that as the yaw angle increases from 0° to 15° yaw, the vortex experiences a size increment of 330 mm in the horizontal component and 60 mm in the vertical component. For the SL model, it can be seen that the vortex is larger in the horizontal component (440 mm at 15° yaw). The horizontal component also experiences a higher increase with an increase in yaw angles (Table 4.4).

Table 4.5: SL Model Vortex Size at 40% scale

<table>
<thead>
<tr>
<th>Yaw Angles</th>
<th>0°</th>
<th>5°</th>
<th>10°</th>
<th>15°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Horizontal (mm)</td>
<td>110</td>
<td>219</td>
<td>310</td>
<td>440</td>
</tr>
<tr>
<td>Vertical (mm)</td>
<td>360</td>
<td>330</td>
<td>320</td>
<td>380</td>
</tr>
</tbody>
</table>

The vortex core angle axis measured relative to the horizontal plane suggests that as the yaw angle increases, the vortex core angle axis generally tend to increase. This finding was consistent with the circular and SL model (Table 4.4). The RE model exhibits an opposite trend to this finding (Table 4.3). Reason for this behaviour was due to the slant angle of the windshield for the RE model. The windshield of the RE model is 90° to the horizontal plane and not at 60° slant as per the other models. This difference in windshield slant angle resulted in a decreasing vortex spread as the yaw angle increases.
It can be seen from Table 4.5 that the circular and the SL model experienced an increase of 5° and 15° respectively when subjected to a 15° increase in yaw angle.

Table 4.6: Model vortex size increase with respect to the horizontal plane

<table>
<thead>
<tr>
<th>Models</th>
<th>0° yaw</th>
<th>15° yaw</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular</td>
<td>5°</td>
<td>10°</td>
</tr>
<tr>
<td>SL</td>
<td>30°</td>
<td>45°</td>
</tr>
<tr>
<td>RE</td>
<td>55°</td>
<td>38°</td>
</tr>
</tbody>
</table>

The results obtained from the CFD analysis shows that for both the circular and sharp edge models, the source of vortex separation behind the A-pillar region originated from the junction of the A-pillar base, the A-pillar apex and the front side window and roof junction. This observation was in agreement with Hanaoka et al. (1993), Ahmed (1998) and Zhu et al. (1993, 1994).

The flow separation behind the A-pillar region was a mixture of two dimensional and three dimensional flows in which the determining factor of the flow mechanism lies in the geometrical configuration of the source. Vortex originated from the A-pillar base junction and the roof junction is two-dimensional free trailing helical vortices. Vortex originated from the A-pillar edge/apex is either two-dimensional free trailing helical vortices or three-dimensional helical vertically elongated cone shape vortices. This observation is in agreement with Alam et al. (1998), Bearman et al. (1989), Haruna et al. (1990) and Barnard (1996).

The mechanism of flow separation for both the circular and sharp edge models was due to trailing edge separation in which the airflow that impinges on the windshield surface started off as laminar flow. The laminar effect causes airflow to slide over each other due to skin friction drag formed with the vehicle wall surface. This will cause the outer air layer moving faster than the inner one which will slow down the flow, causing boundary layer to gradually become thicker. As the formation of turbulent boundary layer
progressed, the flow becomes streamlined, following the contours of the vehicle body. This was in agreement with Haruna et al. (1990) and Barnard (1996).

As the flow separates from the trailing edge, it was observed that region of adverse pressure gradient was formed due to the rapid pressure change between layers of airflow. This was in agreement with Roberson et al. (1997) and Hucho (1998). Viscous slipping between adjacent layers of fluid molecules forces high air velocity flow on the outer layer to glide pass the inner layer formation, causes airflow to circulate, which forms the basis of vortex formation, or ‘dead water’ zone, Hucho (1998). The area between region of flow separation and reattachment is defined as ‘separation bubble’ region, Hucho (1998). According to Watanabe et al. (1978) and Alam (2000) the vortex development forms region of low relative pressure and that the core is located at the vortex core of the separation. The shape of vortex formation varies between the circular and sharp edge models.

For circular models, the shape of the vortices that takes place at $0^\circ$ yaw took a physical form of a two-dimensional quasi-elongated oval with a direction of flow moving downstream to the flow. At $15^\circ$ yaw, the shape of the vortices for the circular models took an overall physical form of a three-dimensional mixture of a quasi circular and cone shaped helical vortex, which was a chaotic combination of two-dimensional and three dimensional vortices originating from the A-pillar base junction, apex and roof junction. The axes of these vortices run essentially in the stream wise direction. The vortex is very rich in kinetic energy and this containment in kinetic energy, which was determined by the vehicle A-pillar angle inclination, Hucho (1998).

For the RE sharp edge model, shape of the localised vortices that takes place at $0^\circ$ yaw was a two-dimensional free trailing vortex originated from the base of the A-pillar junction and a three-dimensional quasi circular cone shape helical vortex generated from the A-pillar apex. The vortex generated from the roof junction formed a three-dimensional circular shape vortex. All three form of vortices flowed in stream wise direction, downstream to the flow. For the RE shape edge model at $15^\circ$ yaw, the shape of
the vortex that takes place at the A-pillar base junction and A-pillar apex was similar to that at 0° yaw. The vortices formed are very rich in kinetic energy, which propagated downstream to the flow, continuing as free trailing vortices, Hucho (1998). The perpendicular angle between the bonnet and windshield, also the windshield and the roof causes the vortices to interfere with each other creating a chaotic turbulent structure. Coupled with the yaw angle, the separation causes turbulent mixing, which translate to a larger separation bubble and enhanced turbulent intensity. The physical shape of the vortex for the RE model at yaw angles are similar to those of the circular models. However, the turbulent intensity are higher in the RE model due to the influence of the sharp A-pillar edge as compared to the streamlines curved A-pillar in the circular models.

For the SL sharp edge model, shape of the cortex that takes place at 0° yaw took a physical form of a three-dimensional vertically elongated cone shape helical vortex propagating downstream to the flow. At 0° yaw and 15° yaw, the shape of the vortex at the A-pillar base junction, the A-pillar slant edge and the roof junction combine to form the three dimensional conical helical vortex, making the turbulent intensity stronger as the yaw angle increases. This was in agreement with Bearman et al. (1989). Changing these parameters affects the intensity, size and shape of the vortex generated behind the A-pillar region, Popat (1991). As the vortices propagates downstream to the flow, turbulent mixing dissipates most of their kinetic energy making their development as continuing free trailing vortices, often weak and even untraceable, Hucho (1998). Turbulent intensity in the SL model is higher compared to the RE and circular models respectively.

In the vortex structure, it was observed that the separated flow within the vortex region exhibit rotational flow while the attached region remains irrotational, which was in agreement with Haruna et al (1990). The small scale turbulent eddies are causing rapid mixing. However, very close to the surface within a turbulent boundary layer flow, a thin sub layer of laminar flow still exists. This result in a gradual increase in relative pressure as the airflow moves downstream, which will have a direct impact on the process of turbulent mixing. It will allow energy transfer to take place from the fast moving eddies
to the slower moving eddies in the turbulent boundary layer resulting in turbulent boundary layer with low turbulent activity.

In a yaw angle scenario, it was observed that strong sideways or cross-stream components on the surface of a vehicle complicate the formation and behaviour of the boundary layer. Cross-stream components are more inclined to cause early transition of the turbulent boundary layer. Cross-stream flows can also keep the boundary layer attached by reducing high-pressure flow, making the pressure gradient less adverse. The complex turbulence behaviour behind the A-pillar apex causes the vortex formed to stretch and breakdown experiencing a decrease in magnitude intensity in the direction normal to wall surface of the model and downstream to the flow. This is in line with the findings of Laufer (1974) and Hussain (1983, 1986).

Overall, observation of various shapes of windshield radii and slant angle models simulated in this project showed that the various geometrical configurations of the windshield radii and slant angle governs the behaviour pattern of vortex generation behind the A-pillar region. This was in agreement with Popat (1991), Hamel (1996) and Hucho, (1998). The observations have showed that the vortex generated behind the A-pillar region was lower in intensity and size in the circular models as compared to the sharp edge models. The circular shape models windshield geometrical configuration and slant angle created three different vortex formations, which determined the direction of flow. The vortex generated from the various sources created a scenario whereby the airflow path interferes with each other, creating a chaotic quasi-circular vortex structure that prevents the formation of highly intense vortices behind the A-pillar region. This scenario has the possibility to limit the generation of aerodynamic noise generated behind the A-pillar region. The sharp edge models windshield geometrical configuration and slant angle created a scenario whereby the airflow path from various directions converges together effectively to form a highly intense three-dimensional helical cone shape vortex all the way downstream to the flow. This produces high velocity region within the vortex, which translate to a region of low negative pressure causing a vacuum like effect, which was in agreement with Watanabe et al. (1978) and Alam (2000). When subjected to an
increased yaw angle, the vortex intensity, magnitude and size increases. This is due to the formation of early boundary layer transition as a direct result from the change in airflow angle of attack on the vehicle surface prior to separation. However again, this is dependent on the geometrical configuration of the windshield radii and slant angle. The turbulent intensity of the circular models at yaw angles is smaller compared to the sharp edge models.
Chapter Five
COMPUTATIONAL AEROACOUSTICS (CAA) SIMULATIONS

This chapter is divided into five sections. The first two section begins with the introduction and methodology of the hybrid SWIFT CAA approach. The third section describes the objectives and scope of this chapter. The fourth and fifth section will be the presentation of results and discussion obtained through the SWIFT CAA simulation.

5.1 Introduction to the Hybrid SWIFT CAA Approach

As described previously in Chapter three, the CAA approach used for this research project was based on the hybrid approach introduced by AVL/TNO through their CAA SWIFT code, which was an extension to their CFD SWIFT code. In reiterating from Chapter three, the main benefits and flexibility of using the hybrid CAA SWIFT approach to determine aerodynamic acoustic propagation of a certain domain was to bypass the necessity of:

- Generating computational domain with extremely large total grid count that extends to the far field range in order to resolve the spatial and temporal scales.

- Conducting transient calculation using extremely small time scale that requires a large amount of parallel computing resources.

- Modelling a large part of the upstream flow of the interested area in order to generate an accurate noise prediction.
These three points outlined the issues that need to be dealt with when conducting CAA simulation by using either the Large Eddy Simulation (LES) or Direct Numerical Simulation (DNS).

In using the approach introduced by AVL/TNO through the hybrid CAA SWIFT, the need for high-density grid generation and transient calculation for determining the aerodynamic noise propagation is eliminated. The hybrid CAA SWIFT approach can be categorised into three steps.

The first step consists of interpolation mapping and transferring of three-dimensional data from the CFD RANS simulation to a CAA domain, meshed with unstructured tetrahedral grid of low density. The second step consists of determining a time accurate acoustic source term. In doing this, hybrid CAA SWIFT uses a statistical model, which was developed by Bechera, 1996 and Longatte, 1998. The statistical model uses statistical turbulent quantities obtained from the CFD RANS simulation (turbulent kinetic energy, eddy length scale and decay rate) in generating turbulent fluctuations, which was needed to determine the acoustic source term. Once the acoustic source term has been determined, the final step was to determine the aerodynamic noise propagation (source to receiver). This was conducted using the Linearized Euler Equation (LEE) and the Discontinuous Galerkin formulation, AVL (2003).
5.2 Methodology of the Hybrid SWIFT CAA Approach

The first step in the hybrid SWIFT CAA approach was to use the CAA Mapper to interpolate and map three-dimensional statistical turbulent quantities from the CFD domain to the CAA domain. In order to initialise and employ the CAA Mapper, the CAA domain however, must first be created.

The CAA domain creation involves a five-step process. The first step in creating the CAA domain was to select and extract an area of interest from within the CFD domain. In the context of this research project, the area of interest consists of the surrounding section containing the A-pillar. The second step involves creating a triangulate surface area from the selected CFD domain extraction using the Triangulation function in the AVL SWIFT Surface Tool section. The third step was to mesh the triangulate surface area with unstructured tetrahedral grids by using the FAME-TET function in the AVL SWIFT Mesh Tool section. The fourth step involves inspecting the newly created unstructured tetrahedral mesh to ensure that it was void of negative volume and negative normal. In addition, each grid cell in the CAA domain must also adhere to an aspect ratio of smaller than 3.3 in order to maintain optimum accuracy. Should problems occur due to the presence of negative volume or normals or due to high value of aspect ratio, changes can be made through altering the values within the FAME-TET function. The final step in creating the CAA domain was to assign boundary conditions to each external surface of the volume. External surface boundaries can be assigned with either a reflecting (wall of the CAA domain) or non-reflecting (inlet and outlet of the CAA domain) boundary condition. The name assigned to the reflecting boundary condition must start with the letter “W” and the name assigned to the non-reflecting boundary condition must start with the letter “N”.

However, simplified vehicle model used in this project especially with rounded windshield configuration and with yawed orientation, creating unstructured tetrahedral grids for the CAA domain was proved to be a difficult exercise since the models experienced a resultant aspect ratio of larger than 3.3. To overcome this problem, the
unstructured tetrahedral was created instead using GAMBIT, from FLUENT. In GAMBIT, the geometry of the CAA domain of interest was first created and then meshed with unstructured tetrahedral grids. The file from GAMBIT was then exported to FLUENT before being saved in a Nastran .BDF file format. The Nastran file was then finally imported using AVL SWIFT and saved as an AVL SWIFT (.FLM) volume file. For the unstructured tetrahedral grids created using the AVL SWIFT functions, each grid size was of 50 mm interval. For the tetrahedral mesh created from GAMBIT, each grid size is of 100 mm interval. Examples of unstructured tetrahedral CAA domain created from AVL SWIFT and GAMBIT can be seen in Figure 5.1.

Figure 5.1 Unstructured Tetrahedral CAA Domain of Various Simplified Vehicle Model Created from AVL SWIFT and GAMBIT
The final mesh count from the unstructured tetrahedral grids generated for the CAA domain varies from one simplified vehicle model to the other. Factors that determined the final mesh generation were the yaw angle orientation of the CAA domain and also the different approach in softwares selection used in creating the CAA domain. Unstructured tetrahedral mesh generated using AVL SWIFT varied in between 14,000 to 35,000 grid cells. Mesh generated using GAMBIT varies in between 5,000 to 17,000 grid cells.

Once the CAA domain (unstructured tetrahedral mesh) has been constructed, the CAA Mapper was then used to map the three-dimensional statistical turbulence quantities from the CFD simulation through the process of interpolation. Dividing the CAA domain into several rectangular partitions or bins accelerates the mapping and interpolation process. The user has to specify the amount of bin in the X, Y and Z direction with a maximum total allowable of 150 bins ($X*Y*Z$). In this research project, the amount of allocated bins was set as default. In addition, the amount of CFD nodes that are being interpolated in each bin must be sufficiently high to ensure an accurate mapping. The user was allowed to set an interpolation value of between 5 to 20 CFD nodes per CAA node to be mapped in each bin. In this research project, the node interpolation value was set as 10, which was the default value. Once the mapping and interpolation process was completed, the CAA Mapper was then able to establish an initial time step to be used in the CAA solver. The initial time step estimate was based on the velocity used in the calculation together with the grid size and the Courant number. It was therefore important to establish the most efficient mesh count in order to obtain a reasonable time step interval for the final simulation in the CAA solver.

As with the final resulting grid size of the unstructured grids, the final resulting time interval for the CAA domain varied from one simplified vehicle model to the other. Unstructured tetrahedral grids generated using AVL SWIFT varied between 4.0 to 6.0 μ-seconds and took between 160,000 to 200,000 time steps to complete an overall 1.0-second time interval propagation. Grids generated using GAMBIT on the other hand varied between 3.0 to 16.0 μ-seconds and will take between 60,000 to 270,000 time steps to complete an overall 1.0-second time interval propagation. The time interval
propagation used for this project was for only 1.0 second as oppose to 10.0 seconds by Alam (2000). Further explanation on this selection will be provided in the following section.

The second and final step in the hybrid SWIFT CAA approach was to use the CAA solver to determine the time accurate acoustic source terms and to finally determine the aerodynamic noise propagation by using the Linearized Euler Equation (LEE) and the Discontinuous Galerkin formulation.

In order to first determine the time accurate acoustic source terms, the workings of the fundamental equations must first be understood. Starting from the derivation of conserved equations of mass, momentum and energy (equations 3.161 to 3.164), it was established that five unknowns exists [density, velocity (three components) and pressure]. The five unknowns are then separated into its mean, acoustic and turbulent component as per equations 3.165 to 3.167. By using LEE, the separated components of density, pressure and velocity was then substituted into the conserved equations of mass, momentum and energy. From there, further derivation took place which in the end produced three acoustic source terms (equations 3.177 to 3.179).

In order to determine the acoustic source term, the turbulent velocity must first (which exists as an unknown) need to be ascertained. This was done through the statistical model developed by Bechera (1996) and Longatte (1998). The statistical model used information from the statistical turbulent quantities obtained from the CFD RANS simulation (turbulent kinetic energy, eddy length scale and decay rate) to generate turbulent fluctuations, which were needed to determine the acoustic source term. The acoustic source term was generated through the Unstructured Kinematic Source Generator (UKSG) in the CAA solver. According to AVL (2003), turbulent velocities were determined through the use of inverse Fourier transform. The inverse Fourier transform distributes the turbulent velocities as superposition of modes. Each inverse Fourier transform mode was assigned with a mode amplitude, random frequency and random phase. The mode amplitude was determined through the use of a modified Von
Karman energy spectrum (which assumes to be incompressible and isotropic) in which rely on the turbulent kinetic energy and dissipation length scale results from the CFD RANS simulation. The inverse Fourier transform formulation was then completed by adding a temporal component to the inverse Fourier transform, in which the spatial and temporal coupling between the turbulence and the mean flow was conducted through the introduction of a moving axis spectrum. Finally, the acoustic source term was then obtained through the differentiation of the Lighthill Stress Tensor components. These source terms are calculated for each element in the CAA domain.

The final step in the CAA solver is to determine aerodynamic acoustic propagation through solving the LEE (equations 3.189 to 3.195). This was done from within the CAA solver by the DIGS3D subroutine. According to AVL (2003), this calculation was conducted in the time domain by using a Quadrature Free Discontinuous Galerkin Spatial discretization approach. In this approach, the revised source term obtained progressively at each time interval from the UKSG subroutine was interpolated to the CAA solver. A new time step interval was then obtained through the use of a fourth-order Runge-Kutta algorithm.

In setting up the CAA solver to simulate for the aerodynamic acoustic propagation, most of the input parameter in the Solver Steering File (Case File) was set as default. The input parameters that were revised were the Turbulence Realization Sampling Frequency and the Input Coordinates for Microphone Location. The values assigned for the Turbulence Realization Sampling Frequency was 40,000 Hz, which was the maximum allowed. This was close to the value of 48,000 Hz, which was used by Alam (2000). The input coordinates for the microphone locations was assigned for all 16 points for the bottom and top row along the allocated area behind the A-pillar region. For yaw cases, coordinates were assigned to A-pillars in the windward and leeward region.

Once the input in the Solver Steering File has been finalised, the CAA solver will process the calculation for the aerodynamic acoustic propagation. The duration of each simulation varies with each simplified vehicle model. Depending on the size of the CAA domain
(total mesh) and time step interval, the simulation took between one to four days to finish. This is also dependent on the computational processing capabilities. Calculation of the CAA solver for this project was done using only a single processor in the Swinburne University Super Computer cluster. Once the CAA solver has finished processing, the data was analysed using post-processing features available in AVL SWIFT. Post processing features such as Power Spectral Density (PSD) analysis were obtained by providing input such as the number of Fourier Fast Transform (FFT) blocks, Number of FFT overlap points and viewing Window for each Fourier block. For this research project, the inputs were obtained from Alam (2000) which were as follows:

- Number of FFT blocks – 4096
- Number of FFT overlap points – 50%
- Viewing Window for each block – Hanning

5.3 Objectives & Scope of using Hybrid SWIFT CAA Approach: Application to this Research Project

There are two main objectives in the context of applying the CAA approach to this research project. The first main objective is to validate the Hybrid SWIFT CAA approach in modelling aeroacoustics behaviour of airflow around the vehicle A-pillar region numerically through the application of the CFD modelling results obtained. The choice of using the Hybrid SWIFT CAA approach was based on the simple approach that it promised, which was discussed earlier in section 5.1. Validation of the Hybrid SWIFT CAA approach in this chapter will be carried out through comparison of results with results obtained experimentally by Alam (2000).

The second main objective is to extend the knowledge on the study of aeroacoustics behaviour of airflow around the A-pillar region, possibly linking it to the aerodynamics behaviour, which was presented and discussed in Chapter 4. From the knowledge
obtained while researching for this project through reviews of literatures (Chapter 2), the following questions were raised and will be addressed to in this chapter.

- In the context of transient flow condition of airflow around the A-pillar region, when does each simplified vehicle model approach steady state condition? Will different yaw orientation have any effect on this?

- What are the aero acoustics behaviour and distribution patterns that can be established in terms of OASPL magnitude at steady and transient condition of different simplified vehicle model at different yaw orientation?

- How is the aero acoustics behaviour behind the A-pillar region linked with the aerodynamics behaviour at different simplified vehicle model at different yaw orientation?

Therefore, the objectives along with the scope of this chapter will be as follows:

**5.3.1 Objectives of Chapter 5**

1. To validate the Hybrid SWIFT CAA modelling of the aero acoustics of airflow behaviour surrounding the vehicle A-pillar region with the existing experimental results of Alam (2000).

2. To determine the temporal transition between transient and steady state flow condition of airflow around the A-pillar region at different yaw orientation.

3. To determine the behaviour and distribution mode of OASPL magnitude at steady state and transient condition surrounding the A-pillar region within the vicinity of the CAA domain at different yaw orientation.
4. To link the aero acoustics and aerodynamic behaviour behind the A-pillar region for at different yaw orientation.

5.3.2 Scope of Chapter 5

1. All simplified models (RE (Rectangular), SL (Slanted Edge), SE (Small Ellipsoidal), Semi (Semi Circular), and LE (Large Ellipsoidal)).

2. Simplified model yaw orientation of 0° and 15°.

3. Wind tunnel inlet velocity of 60, 100 and 140 km/h. For objectives number two to five, wind tunnel inlet velocity of only 140 km/h will be used.

4. Time interval of 1.0 second.
5.4 Hybrid SWIFT CAA Results

The results obtained from using hybrid SWIFT CAA modelling will be presented in this section.

5.4.1 Hybrid SWIFT CAA & Experimental Validation - SE Model, 0° & 15° Yaw

Comparison of $C_p$ RMS values between CAA and experimental results for the SE model at 0° yaw can be seen in Figure 5.2. Good correlation was obtained from both results for both the bottom and top row monitoring locations. The maximum and minimum $C_p$ RMS values obtained from CAA simulation were 0.038 and 0.015 respectively. The maximum and minimum $C_p$ RMS values obtained experimentally by Alam (2000) were 0.04 and 0.025 respectively.

![Figure 5.2: Comparison of Cp RMS between Numerical and Experimental Results, Small Ellipsoidal Model, 0° Yaw](image_url)
Comparison of $C_p$ RMS values obtained for the SE model at 15° yaw between the CAA simulation and experimental results can be seen on Figure 5.3. It can be seen from Figure 5.3 that correlation between experimental and CAA simulation results were similar in the windward region for both the bottom and top row monitoring locations. However, small discrepancies exist in the leeward region, particularly in the halfway point of the bottom row monitoring locations and towards the end of the top row monitoring locations respectively, Murad (2006). The maximum $C_p$ RMS obtained for the SE Model at 15° yaw at the leeward and windward region from the CAA simulation was 0.048 and 0.039 respectively. The minimum $C_p$ RMS value that was obtained at the leeward and windward region was 0.027 and 0.022 respectively. The maximum $C_p$ RMS value in the leeward and windward region obtained experimentally by Alam (2000) was 0.13 and 0.045. The minimum $C_p$ RMS value in the leeward and windward region obtained experimentally by Alam (2000) was 0.04 and 0.035 respectively.

The comparison of PSD distribution between CAA simulation and experimental results obtained for SE model of 0°, -15° and +15° yaw angle at 100 km/h can be seen in Figure 5.4. From the CAA simulation results in Figure 5.4, it can be seen that the highest prediction of PSD distribution occurred at -15° yaw, followed by +15° and 0° yaw.
respectively. The peak OASPL value obtained from the CAA simulation results at -15° yaw was measured at 99-dB followed by +15° yaw measured at 98-dB. Finally, peak OASPL value obtained from CAA results at 0° yaw was measured at 100-dB. Results obtained experimentally from Alam (2000) yielded peak OASPL value at -15° yaw measured around 113-dB, followed by a measurement of 101-dB at 0° yaw (Figure 5.7). Finally a measurement of 95-dB was obtained at +15° yaw orientation. Difference between peak OASPL values between CAA and experimental results yielded lowest discrepancies at 0° yaw, followed by +15° yaw and -15° yaw, measuring 1.0-dB, 3.0-dB and 14.0-dB respectively, Murad (2006).
Both the experimental and CAA PSD results showed an intersection at 3-kHz (Figure 5.6). Although the experimental results showed only a single spectral energy cross over at 3-kHz, the CAA simulation results from Figure 5.6 showed an additional cross over at 500-Hz and 6-kHz (Figure 5.5) respectively. On both occasion, PSD fluctuation was
observed originating from the model at 0° yaw angle. The overall PSD distribution discrepancy below 3-kHz, was measured at 1.0-dB, 5.5-dB and 12.5-dB, corresponding to yaw angles at +15°, 0° and -15° respectively. The overall PSD discrepancies obtained above 3-kHz was 9.0-dB, 9.0-dB and 14.0-dB, corresponding to yaw angles at +15°, 0° and -15° respectively. In summary, the overall PSD distribution for SE model showed best correlation at yaw angle of +15°, followed by yaw angles of 0°-yaw and -15°-yaw, with discrepancies (combining discrepancies for below and above 3-kHz) measured at 5.0-dB, 7.3-dB and 13.3-dB respectively. Discrepancies in the CAA results is caused by the discrepancies obtained in the earlier CFD models (discussed in Chapter 4) and also due to numerical discrepancies caused by the acoustical model used in software to determine the acoustical propagation estimation.

5.4.2 Hybrid SWIFT CAA & Experimental Validation - Semi Model, 0° & 15° Yaw

Comparison between experimental and CAA results of $C_p$ RMS values for the Semi model at 0° yaw can be seen in Figure 5.7. Good correlation was obtained from both results for both the bottom and top row monitoring locations. The maximum and minimum $C_p$ RMS values obtained from the CAA simulation were 0.067 and 0.044 respectively. The maximum and minimum $C_p$ RMS values obtained experimentally by Alam (2000) were 0.05 and 0.025 respectively. This discrepancy in results is small and acceptable due to the constant fluctuating of acoustical pressure prior conversion to $C_p$ RMS values.

The $C_p$ RMS results for the Semi model at 15° yaw obtained from the CAA simulation can be seen on Figure 5.8. It can be seen from Figure 5.8 that correlation between experimental and CAA simulation results in the leeward and windward region yielded small discrepancies, particularly towards the rear portion of the bottom and top row monitoring points.
Figure 5.7: Comparison of Cp RMS between Numerical and Experimental Results, Semi Circular Model, 0° Yaw

Figure 5.8: Comparison of Cp RMS between Numerical and Experimental Results, Semi Circular Model, -15° Yaw
The maximum $C_p$ RMS values obtained for the Semi model at 15° yaw at the leeward and windward region from the CAA simulation were 0.035 and 0.034 respectively. The minimum $C_p$ RMS values that were obtained at the leeward and windward region were 0.022 and 0.020 respectively. The maximum $C_p$ RMS values obtained experimentally by Alam (2000) for the leeward and windward region were 0.08 and 0.04 respectively. In addition, the minimum $C_p$ RMS values obtained experimentally at the leeward and the windward region were 0.04 and 0.025 respectively. Again, as with the 0° yaw model, this discrepancy in results is small and acceptable due to the constant fluctuating of acoustical pressure prior conversion to $C_p$ RMS values.

![Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Semi Circular Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 1.4-kHz Frequency Region](image)

Figure 5.9: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Semi Circular Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 1.4-kHz Frequency Region
The comparison of PSD distribution between CAA simulation and experimental results obtained for Semi model of 0°, -15° and +15° yaw angle at 100 km/h can be seen in Figure 5.9. From the CAA simulation results in Figure 5.9, it can be seen that throughout the PSD distribution, the model at 0° yaw experienced the highest prediction of PSD distribution. These were followed by +15° and -15° yaw respectively. The experimental results produced the highest PSD distribution at the -15° yaw model, followed by the 0° yaw and +15° yaw respectively (Figure 5.10).

From the CAA simulation results, the peak OASPL value obtained at 0° yaw was measured at 103-dB followed by -15° yaw measured at 102-dB. Finally, maximum OASPL value obtained at +15° yaw was measured at 98-dB. Results obtained experimentally from Alam (2000) yielded peak OASPL value at -15° yaw measured around 120-dB, followed by a measurement of 117-dB at 0° and +15° yaw respectively. Comparison of peak PSD values between CAA and experimental results yielded lowest discrepancy at 0° yaw, followed by -15° and +15° yaw, measuring 14.0-dB, 18.0-dB and 19-dB respectively, Murad (2006).
Overall, the PSD distribution prediction was at its most consistent for model at 0°, followed by models at +15° and -15° yaw respectively. These corresponds to mean average discrepancies (combining discrepancies at peak and minimum OASPL) of 7-dB, 12-dB and 15-dB in magnitude for yaw angle of 0°, +15° and -15° yaw respectively.

5.4.3 Hybrid SWIFT CAA & Experimental Validation - LE Model, 0° & 15° Yaw

Comparison between experimental and CAA results of \( C_p \) RMS values for the LE model at 0° yaw can be seen in Figure 5.11. Good correlation was obtained from both results for both the bottom and top row monitoring locations. However, comparison of results showed small discrepancy at 0° yaw and 100 km/h, caused probably by discretization error. The maximum and minimum \( C_p \) RMS values obtained from the CAA simulation were 0.063 and 0.021 respectively. The maximum and minimum \( C_p \) RMS values obtained experimentally by Alam (2000) were 0.040 and 0.08 respectively.

The \( C_p \) RMS results for the LE model at 15° yaw obtained from the CAA simulation can be seen on Figure 5.12. It can be seen from Figure 5.12 that correlation between experimental and CAA simulation results were similar in the leeward and windward region for both the bottom and top row monitoring locations. However, small discrepancies exist around the frontal portion of the bottom and top monitoring locations of the windward region. The maximum \( C_p \) RMS values obtained for the LE model at 15° yaw at the leeward and windward region from the CAA simulation were 0.044 and 0.033 respectively. The minimum \( C_p \) RMS values that were obtained at the leeward and windward region were 0.026 and 0.045 respectively. The maximum \( C_p \) RMS values obtained experimentally by Alam (2000) for the leeward and windward region were 0.046 and 0.045 respectively. The minimum \( C_p \) RMS values obtained experimentally at the leeward and the windward region was 0.121 and 0.067 respectively.
Figure 5.11: Comparison of Cp RMS between Numerical and Experimental Results, Large Ellipsoidal Model, 0° Yaw

Figure 5.12: Comparison of Cp RMS between Numerical and Experimental Results, Large Ellipsoidal Model, -15° Yaw
The comparison of PSD distribution between CAA simulation and experimental results obtained for LE model of 0°, -15° and +15° yaw angle at 100 km/h can be seen in Figure 5.13. From the CAA simulation results in Figure 5.13, it can be seen that the highest prediction over the majority of the PSD distribution was produced at 0° yaw, followed by -15° and +15° yaw respectively. The experimental results produced the highest order of PSD distribution at -15° yaw model, followed by the +15° and 0° yaw respectively (Figure 5.15).

From the CAA simulation results, the peak OASPL value obtained at 0° yaw was measured at 105-dB followed by -15° yaw measured at 102-dB. Finally, peak OASPL value obtained at +15° yaw was measured at 99-dB. Results obtained experimentally from Alam (2000) yielded peak OASPL value at -15° yaw measured around 115 dB, followed by a measurement of 114-dB and 110-dB at +15° and 0° yaw respectively. Discrepancies of around 13-dB, 15-dB and 5-dB were obtained at -15°, +15° and 0° yaw when comparing the peak OASPL values obtained from CAA simulation and experimental results.

![Power Spectral Density (PSD) Distribution](image)

Figure 5.13: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Large Ellipsoidal Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 1.2-kHz Frequency Region
Cross Over at 2.2-kHz

Cross Over at 5.2-kHz

Figure 5.14: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Large Ellipsoidal Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8.0-kHz Frequency Region

Figure 5.15: Experimental Results of Spectral Energy Density Distribution for Large Ellipsoidal Model at -15°, 0° and +15° Yaw (After Alam, 2000)
The CAA simulation however, did not reproduce the intersection between PSD content that was occurring at around 800-Hz as showed evidently from the experimental results. The CAA simulation results from Figure 5.14 showed a PSD cross over at 2.2-kHz and 5.2-kHz respectively. Both crossovers were caused by a gradual decrease in spectral energy content for the model at 0° yaw angle.

Overall, it can be seen from the CAA simulation results that the prediction of PSD was at its most consistent for model at 0°, followed by model at -15° and +15° yaw respectively. The overall mean average discrepancies (combining discrepancies at peak and minimum OASPL) for PSD distribution of LE model was 3-dB, 6.75-dB and 10-dB for yaw angle of 0°, -15° and +15° yaw respectively, Murad (2006).

### 5.4.4 Hybrid SWIFT CAA & Experimental Validation - RE Model, 0° & 15° Yaw

Comparison between experimental and CAA results of $C_p$ RMS values for the RE model at 0° yaw can be seen in Figure 5.16. Good correlation was obtained from both results for both the bottom and top row monitoring locations. However, small discrepancies exist around the front section of the bottom row monitoring location and around the middle section of the top row monitoring locations. The maximum and minimum $C_p$ RMS values obtained from the CAA simulation were 0.231 and 0.114 respectively. The maximum and minimum $C_p$ RMS values obtained experimentally by Alam (2000) were 0.23 and 0.08 respectively.

The $C_p$ RMS results for the RE model at 15° yaw obtained from the CAA simulation can be seen on Figures 5.17 and 5.18. It can be seen from Figures 5.17 and 5.18 that correlation between experimental and CAA simulation results yield discrepancies for the bottom and top row monitoring locations in the leeward and windward region. The maximum $C_p$ RMS values obtained for the RE model at 15° yaw at the leeward and windward region from the CAA simulation were 0.187 and 0.148 respectively. The
minimum $C_p$ RMS values that were obtained at the leeward and windward region were 0.098 and 0.067 respectively. The maximum $C_p$ RMS values obtained experimentally by Alam (2000) for the leeward and windward region were 0.33 and 0.4 respectively. The minimum $C_p$ RMS values obtained experimentally at the leeward and the windward region was 0.07 and 0.04 respectively.

Figure 5.16: Comparison of $C_p$ RMS between Numerical and Experimental Results, Rectangular Edge Model, 0° Yaw
Figure 5.17: Comparison of Cp RMS between Numerical and Experimental Results, Rectangular Edge Model, -15° Yaw

Figure 5.18: Comparison of Cp RMS between Numerical and Experimental Results, Rectangular Edge Model, +15° Yaw
The comparison of PSD distribution between CAA simulation and experimental results obtained for RE model of 0°, -15° and +15° yaw angle at 100 km/h can be seen in Figure 5.19. From the CAA simulation results in Figure 5.19, it can be seen that the prediction of highest order of PSD distribution occurred at 0° yaw, followed by -15° and +15° yaw respectively. It can be seen from Figure 5.19 that the CAA results produced a PSD cross over at 4000 Hz. The cross over was caused by a gradual decrease in spectral energy content for the model at -15° yaw angle.

From the CAA simulation results, the peak OASPL value obtained at 0° yaw was measured at 117-dB followed by -15° yaw measured at 116-dB. Finally, peak OASPL value obtained at +15° yaw was measured at 109-dB. Results obtained experimentally from Alam (2000) yielded a peak OASPL value at -15° yaw measured around 126.5-dB, followed by peak values of 128-dB and 124-dB at +15° and 0° yaw respectively. Discrepancies of 10.5-dB, 19-dB and 7-dB were obtained at -15°, +15° and 0° yaw when comparing the peak OASPL values obtained between CAA and experimental results.

![Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Rectangular Edge Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8.0-kHz Frequency Region](image_url)
Overall, the CAA simulation results prediction of PSD distribution was most consistent for model at 0°, followed by model at -15° and +15° yaw respectively. The overall mean average discrepancies (combining discrepancies at peak and minimum OASPL) of PSD distribution for RE model was 7-dB, 8.3-dB and 13-dB for yaw angle of 0°, -15° and +15° yaw respectively.

### 5.4.5 Hybrid SWIFT CAA & Experimental Validation - SL Model, 0° & 15° Yaw

Comparison between experimental and CAA results of $C_p$ RMS values for the SL model at 0° yaw can be seen in Figure 5.21. It can be seen from Figure 5.21 that correlation between experimental and CAA simulation results yield discrepancies at the bottom and top row monitoring locations. This was particularly evident in both frontal section of the bottom and top row monitoring locations. The maximum and minimum $C_p$ RMS values obtained from the CAA simulation were 0.21 and 0.07 respectively. The maximum and
minimum $C_p$ RMS values obtained experimentally by Alam (2000) were 0.28 and 0.03 respectively. However, similar experimental testing using SL model, which was conducted by Popat (1991), yielded $C_p$ RMS values of a maximum and minimum value of 0.21 and a 0.06 respectively.

The $C_p$ RMS results for the SL model at 15° yaw obtained from the CAA simulation can be seen on Figure 5.22 and 5.23 respectively. It can be seen from Figure 5.22 and 5.23 that comparison between experimental and CAA simulation results yielded discrepancies for the bottom and top row monitoring locations in the leeward and windward region. This was particularly evident in the middle section of the bottom and top row monitoring locations in the leeward region. In the windward region, discrepancies were observed in the frontal section of the bottom and top row monitoring locations. The maximum $C_p$ RMS values obtained for the Slanted Edge model at 15° yaw at the leeward and windward region from the CAA simulation were 0.16 and 0.1 respectively. The minimum $C_p$ RMS values that were obtained at the leeward and windward region were 0.06 and 0.049 respectively. The maximum $C_p$ RMS values obtained experimentally by Alam (2000) for the leeward and windward region were 0.41 and 0.11 respectively. The minimum $C_p$ RMS values obtained experimentally at the leeward and the windward region was 0.08 and 0.02 respectively.
Figure 5.21: Comparison of Cp RMS between Numerical and Experimental Results, Slanted Edge Model, 0° Yaw

Figure 5.22: Comparison of Cp RMS between Numerical and Experimental Results, Slanted Edge Model, -15° Yaw
The comparison of PSD distribution between CAA simulation and experimental results obtained for SL model of 0°, -15° and +15° yaw angle at 100 km/h can be seen in Figures 5.24 and 5.25. From the CAA simulation results, the peak OASPL value obtained at 0° yaw was measured at 112.5-dB followed by +15° yaw measured at 109-dB. Finally, peak OASPL value obtained at -15° yaw was measured at 102.5-dB. Results obtained experimentally from Alam (2000) yielded a peak OASPL value at -15° yaw measured around 129-dB, followed by a measurement of 116-db and 123-dB at +15° and 0° yaw respectively. Discrepancies of 26.5-dB, 7-dB and 10.5-dB were obtained at -15°, +15° and 0° yaw when comparing the peak OASPL values obtained from CAA simulation and experimental results.

The overall PSD distribution prediction from the CAA simulation results was at its most consistent at 0°, followed by model at -15° and +15° yaw respectively. This corresponds to the total mean average discrepancies (combining discrepancies at peak and minimum OASPL) of 15.3-dB, 17-dB and 5.3-dB for SL model at yaw angle of 0°, -15° and +15° yaw respectively.
Figure 5.24: Power Spectral Density (PSD) Distribution of Maximum RMS Pressure, Slanted Edge Model, 0°, -15° and +15° Yaw, 100 km/h, 0 to 8.0-kHz Frequency Region

Figure 5.25: Experimental Results of Spectral Energy Density Distribution for Slanted Edge Model at -15°, 0° and +15° Yaw (After Alam, 2000)
5.4.6 Aeroacoustics Behaviour during Transient Condition - SE Model, 0° & 15° Yaw

Figure 5.26 and 5.27 presents the mean and RMS transient pressure distribution for the SE model at 0° and 15° yaw over a one second time interval march. The pressure monitoring point was plotted at the front section of the bottom row monitoring location of the A-pillar region. The transient progression of the model was divided into three stages, namely the random fluctuating stage, the transition stage and the steady state stage. At the end of the random fluctuating and transition stage, the time mean average at 60, 100 and 140 km/h were taken to determine the time taken for each stage to end. At 0° yaw, the mean average of the random fluctuating stage ended at 0.08 second while the mean average of the transition stage ended at 0.19 second. Therefore, the random fluctuating stage was between 0 and 0.08 second, while the transition stage was between 0.08 and 0.19 second. The flow finally stabilised into a steady state condition at 0.19 second, Murad (2007). At 15° yaw, the mean average of the random fluctuating stage ended at 0.10 second while the mean average of the transition stage ended at 0.58 second. Therefore, the random fluctuating stage was between 0 and 0.10 second, while the transition stage was between 0.10 and 0.58 second. The flow finally stabilised into a steady state condition at 0.58 second, Murad (2007).
Figure 5.26: Cp RMS Temporal Progression for Small Ellipsoidal Model, 0° Yaw, 60, 100 and 140 km/h

Figure 5.27: Cp RMS Temporal Progression for Small Ellipsoidal Model, 15° Yaw, 60, 100 and 140 km/h
5.4.7 Aeroacoustics Behaviour during Transient Condition - Semi Model, 0° & 15° Yaw

Figure 5.28 and 5.29 presents the mean and RMS transient pressure distribution for the Semi model at 0° and 15° yaw over a one second time interval march. At 0° yaw, the mean average of the random fluctuating stage ended at 0.07 second while the mean average of the transition stage ended at 0.16 second. Therefore, the random fluctuating stage was between 0 and 0.07 second, while the transition stage was between 0.07 and 0.16 second. The flow finally stabilised into a steady state condition at 0.16 second, Murad (2007). At 15° yaw, the mean average of the random fluctuating stage ended at 0.06 second while the mean average of the transition stage ended at 0.35 second. Therefore, the random fluctuating stage was between 0 and 0.06 second, while the transition stage was between 0.06 and 0.35 second. The flow finally stabilised into a steady state condition at 0.35 second, Murad (2007).

Figure 5.28: Cp RMS Temporal Progression for Semi Circular Model, 0 Degree Yaw at 60, 100 and 140 km/h

Figure 5.28: Cp RMS Temporal Progression for Semi Circular Model, 0° Yaw, 60, 100 and 140 km/h
5.4.8 Aeroacoustics Behaviour during Transient Condition - LE Model, 0° & 15° Yaw

Figure 5.30 and 5.31 presents the mean and RMS transient pressure distribution for the Large Ellipsoidal model at 0° and 15° yaw over a one second time interval march. At 0° yaw, the mean average of the random fluctuating stage ended at 0.02 second while the mean average of the transition stage ended at 0.13 second. Therefore, the random fluctuating stage was between 0 and 0.02 second, while the transition stage was between 0.02 and 0.13 second. The flow finally stabilised into a steady state condition at 0.13 second, Murad (2007). At 15° yaw, the mean average of the random fluctuating stage ended at 0.04 second while the mean average of the transition stage ended at 0.26 second. Therefore, the random fluctuating stage was between 0 and 0.04 second, while the transition stage was between 0.04 and 0.26 second. The flow finally stabilised into a steady state condition at 0.26 second, Murad (2007).
Figure 5.30: Cp RMS Temporal Progression for Large Ellipsoidal Model, 0° Yaw, 60, 100 and 140 km/h

Figure 5.31: Cp RMS Temporal Progression for Large Ellipsoidal Model, 15° Yaw, 60, 100 and 140 km/h
5.4.9 Aeroacoustics Behaviour during Transient Condition - RE Model, 0° & 15° Yaw

Figure 5.32 and 5.33 presents the mean and RMS transient pressure distribution for the RE model at 0° and 15° yaw over a one second time interval march. At 0° yaw, the mean average of the random fluctuating stage ended at 0.05 second while the mean average of the transition stage ended at 0.23 second. Therefore, the random fluctuating stage was between 0 and 0.05 second, while the transition stage was between 0.05 and 0.23 second. The flow finally stabilised into a steady state condition at 0.23 second. At 15° yaw, the mean average of the random fluctuating stage ended at 0.06 second while the mean average of the transition stage ended at 0.29 second. Therefore, the random fluctuating stage was between 0 and 0.06 second, while the transition stage was between 0.06 and 0.29 second. The flow finally stabilised into a steady state condition at 0.29 second.

Figure 5.32: Cp RMS Temporal Progression for Rectangular Edge Model, 0 Degree Yaw at 60, 100 and 140 km/h

Figure 5.32: Cp RMS Temporal Progression for Rectangular Edge Model, 0° Yaw, 60, 100 and 140 km/h
5.4.10 Aeroacoustics Behaviour during Transient Condition - SL Model, 0° & 15° Yaw

Figure 5.34 and 5.35 presents the mean and RMS transient pressure distribution for the Slanted Edge model at 0° and 15° yaw over a one second time interval march. At 0° yaw, the mean average of the random fluctuating stage ended at 0.09 second while the mean average of the transition stage ended at 0.34 second. Therefore, the random fluctuating stage was between 0 and 0.09 second, while the transition stage was between 0.09 and 0.34 second. The flow finally stabilised into a steady state condition at 0.34 second. At 15° yaw, the mean average of the random fluctuating stage ended at 0.09 second while the mean average of the transition stage ended at 0.50 second. Therefore, the random fluctuating stage was between 0 and 0.09 second, while the transition stage was between 0.09 and 0.50 second. The flow finally stabilised into a steady state condition at 0.50 second.
Figure 5.34: Cp RMS Temporal Progression for Slanted Edge Model, 0° Yaw, 60, 100 and 140 km/h

Figure 5.35: Cp RMS Temporal Progression for Slanted Edge Model, 15° Yaw, 60, 100 and 140 km/h
5.4.11 CAA Behaviour & Distribution behind A-pillar Region
- SE Model, 0° Yaw

The results obtained showed that under steady condition, the OASPL magnitude on the SE model surface at 0° yaw was measured at an average of 122.7-dB. As the aeroacoustics propagates away from the surface, the OASPL experienced a decrease of 2.1-dB. From Figure 5.36, it can be seen that the two-dimensional trailing edge vortex region propagating from the A-pillar base junction was measured at an average of 123.0-dB. The two-dimensional attached turbulent boundary layer flow propagating from the A-pillar apex propagated OASPL was measured at an average of 119.0-dB. Progressing downstream of the flow, it can be seen from Figure 5.36 that the aero-acoustic propagation due to the A-pillar base junction and A-pillar apex vortex remain close to the surface of the vehicle while at the same time propagating sideways away from the A-pillar base junction. From Figure 5.36, plan view of the aero-acoustic propagation shows a reduction of 4-dB moving away from the A-pillar apex and base junction.

Under transient flow conditions, the initial OASPL magnitude on the model surface was measured at an average of 119.5-dB (Figure 5.37). As the aero-acoustics propagation reached steady state, it experienced an increase of 3.2-dB taking it to 122.7-dB. From Figures 5.38 and 5.39 it can be seen that the aero-acoustic activity is continuous propagating from the A-pillar base junction. However, the aero-acoustic propagation from the A-pillar apex reached steady state after 0.1-second.
Figure 5.36: OASPL, Frontal and Surface View, Steady State Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.37: OASPL, Surface View, Transient Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.38: OASPL, Top View (Base), Transient Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.39: OASPL, Top View (Bottom), Transient Condition, Small Ellipsoidal Model, 0° Yaw, 140 km/h
5.4.12 CAA Behaviour & Distribution Behind A-pillar Region - SE Model, 15° Yaw

The results obtained showed that under steady condition, the OASPL magnitude on the SE model surface at 15° yaw was measured at an average of 123.0-dB. As the aero acoustics propagates away from the surface, the OASPL experienced a decrease of 4.5-dB. From Figure 5.40, it can be seen that the two-dimensional trailing edge vortex region propagating from the A-pillar base junction was measured at an average of 125.0-dB. The three-dimensional and three-dimensional quasi-elongated helical from the A-pillar apex propagation was measured at an average of 125.0-dB. From Figure 5.40, plan view of the aero-acoustic propagation shows a reduction of 2.0-dB moving away from the A-pillar apex and base junction.

Under transient flow conditions, the initial OASPL magnitude on the model surface was measured at an average of 121.5-dB in the leeward region and 119.0-dB in the windward region (Figure 5.41). As the aero-acoustics propagation reached steady state, the leeward region experienced an increase of 1.5-dB and the windward region experienced an increase of 3.5-dB. From Figures 5.42 and 5.43 it can be seen that the aero-acoustic activity is continuous propagating from the A-pillar base junction and apex.
Figure 5.40: OASPL, Surface and Top View, Steady State Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.41: OASPL, Surface View (Leeward), Transient Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.42: OASPL, Top View (Base), Transient Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.43: OASPL, Top View (Bottom), Transient Condition, Small Ellipsoidal Model, 15° Yaw, 140 km/h
5.4.13 CAA Behaviour & Distribution Behind A-pillar Region - Semi Model, 0° Yaw

The results obtained showed that under steady condition, the OASPL magnitude on the Semi model surface at 0° yaw was measured at an average of 121.5-dB. As the aero acoustics propagates away from the surface, the OASPL experienced a decrease of 3.4-dB. From Figure 5.44, it can be seen that the two-dimensional free trailing edge vortex region propagating from the A-pillar base junction was measured at an average of 122.5-dB. The two-dimensional attached turbulent boundary layer flow propagating from the A-pillar apex was measured at an average of 117.0-dB. Progressing downstream of the flow, it can be seen from Figure 5.44 that the aero-acoustic propagation due to the A-pillar base junction and A-pillar apex vortex remain close to the surface of the vehicle while at the same time propagating sideways away from the A-pillar base junction experiencing a reduction of 2.0-dB.

Under transient flow conditions, the initial OASPL magnitude on the model surface was measured at an average of 117.2-dB (Figure 5.45). As the aero-acoustics propagation reached steady state, it experienced an increase of 4.3-dB taking it to 121.5-dB. From Figures 5.46 and 5.47 it can be seen that the aero-acoustic activity is continuous propagating from the A-pillar base junction, reaching steady state after 0.1-second. The aero-acoustic propagation from the A-pillar apex reached steady state after 0.02-second.
Figure 5.44: OASPL, Frontal and Surface View, Steady State Condition, Semi Circular Model, 0° Yaw, 140 km/h
Figure 5.45: OASPL, Surface View, Transient Condition, Semi Circular Model, 0° Yaw, 140 km/h
Figure 5.46: OASPL, Top View (Base), Transient Condition, Semi Circular Model, 0° Yaw, 140 km/h
Figure 5.47: OASPL, Top View (Bottom), Transient Condition, Semi Circular Model, 0° Yaw, 140 km/h
5.4.14 CAA Behaviour & Distribution Behind A-pillar Region - Semi Model, 15° Yaw

The results obtained showed that under steady condition, the OASPL magnitude on the Semi model surface at 15° yaw was measured at an average of 122.2-dB. As the aero acoustics propagates away from the surface, the OASPL experienced a decrease of 3.5-dB. From Figure 5.48, it can be seen that the two-dimensional trailing edge vortex region propagating from the A-pillar base junction was measured at an average of 123.0-dB. The three-dimensional and three-dimensional quasi-elongated helical from the A-pillar apex propagation was measured at an average of 123.0-dB. From Figure 5.48, plan view of the aero-acoustic propagation shows a reduction of 3.0-dB moving away from the A-pillar apex and base junction.

Under transient flow conditions, the initial OASPL magnitude on the model surface was measured at an average of 120.5-dB in the leeward region and 118.9-dB in the windward region (Figure 5.49). As the aero-acoustics propagation reached steady state, the leeward region experienced an increase of 2.2-dB and the windward region experienced an increase of 2.8-dB. From Figures 5.50 and 5.51 it can be seen that the aero-acoustic activity is continuous propagating from the A-pillar base junction, reaching steady state condition after 0.05-second and behind the A-pillar apex, reaching steady state condition after 0.2-second.
Figure 5.48: OASPL, Surface and Top View, Steady State Condition, Semi Circular Model, 15° Yaw, 140 km/h
Figure 5.49: OASPL, Surface View (Leeward), Transient Condition, Semi Circular Model, 15° Yaw, 140 km/h
Figure 5.50: OASPL, Top View (Base), Transient Condition, Semi Circular Model, 15° Yaw, 140 km/h
Figure 5.51: OASPL, Top View (Bottom), Transient Condition, Semi Circular Model, 15° Yaw, 140 km/h
5.4.15 CAA Behaviour & Distribution Behind A-pillar Region - LE Model, 0° Yaw

The results obtained showed that under steady condition, the OASPL magnitude on the LE model surface at 0° yaw was measured at an average of 120.8-dB. As the aero acoustics propagates away from the surface, the OASPL experienced a decrease of 4.2-dB. From Figure 5.52, it can be seen that the two-dimensional trailing edge vortex region propagating from the A-pillar base junction was measured at an average of 120.0-dB. The two-dimensional attached turbulent boundary layer flow propagating from the A-pillar apex propagation was measured at an average of 116.5-dB. Progressing downstream of the flow, it can be seen from Figure 5.52 that the aero-acoustic propagation due to the A-pillar base junction and A-pillar apex vortex remain close to the surface of the vehicle while at the same time propagating sideways away from the A-pillar base junction, experiencing a reduction of 1-dB.

Under transient flow conditions, the initial OASPL magnitude on the model surface was measured at an average of 115.8-dB (Figure 5.53). As the aero-acoustics propagation reached steady state, it experienced an increase of 5.0-dB taking it to 120.8-dB. From Figures 5.54 and 5.55 it can be seen that the aero-acoustic activity is continuous propagating from the A-pillar base junction, reaching steady state condition after 0.02-second. However, the aero-acoustic propagation from the A-pillar apex reached steady state after 0.08 second.
Figure 5.52: OASPL, Surface and Frontal View, Steady State Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.53: OASPL, Surface View, Transient Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.54: OASPL, Top View (Base), Transient Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h
Figure 5.55: OASPL, Top View (Bottom), Transient Condition, Large Ellipsoidal Model, 0° Yaw, 140 km/h
5.4.16 CAA Behaviour & Distribution Behind A-pillar Region - LE Model, 15° Yaw

The results obtained showed that under steady condition, the OASPL magnitude on the LE model surface at 15° yaw was measured at an average of 121.6-dB. As the aero acoustics propagates away from the surface, the OASPL experienced a decrease of 3.0-dB. From Figure 5.56, it can be seen that the two-dimensional trailing edge vortex region propagating from the A-pillar base junction and the three-dimensional and three-dimensional quasi-elongated helical from the A-pillar apex propagation was measured at an average of 121.0-dB and 122.5-dB respectively. From Figure 5.40, plan view of the aero-acoustic propagation shows a reduction of 4-dB moving away from the A-pillar apex and base junction.

Under transient flow conditions, the initial OASPL magnitude on the model surface was measured at an average of 120.0-dB in the leeward region and 118.7-dB in the windward region (Figure 5.57). As the aero-acoustics propagation reached steady state, the leeward region experienced an increase of 2.7-dB and the windward region experienced an increase of 2.2-dB. From Figures 5.42 and 5.43 it can be seen that the aero-acoustic activity is continuous propagating from the A-pillar base junction and apex, reaching steady state condition after 0.1-second.

The results obtained from the simulation showed that the acoustical intensity is higher in region closer to the vehicle surface and decreases as it moves away from the surface. Highest acoustical intensity occurs in the A-pillar region. As the vehicle is yawed, acoustical intensity increases with time before it reaches steady state condition. The leeward region, which experience a larger separation compared to the windward region, produces a higher acoustical intensity. Finally, the results show that steady state condition was reached only after 0.1-second. After 0.1-second the overall vortex structure formation has fully evolved. However, subtle turbulence unsteadiness within the vortex structure is will always be present. The behaviour and distribution of the acoustical propagation in the surrounding A-pillar region is a direct consequence to the formation of the A-
pillar vortex due to the geometrical characteristics of the vehicle A-pillar and windshield radii and also due to the exposure to various yaw angle orientation.
Figure 5.56: OASPL, Surface and Top View, Steady State Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.57: OASPL, Surface View (Leeward), Transient Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.58: OASPL, Top View (Base), Transient Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h
Figure 5.59: OASPL, Top View (Bottom), Transient Condition, Large Ellipsoidal Model, 15° Yaw, 140 km/h
5.5 Discussion of Hybrid SWIFT CAA Results

The discussion of results obtained from using hybrid SWIFT CAA modelling will be presented in this section. Discussion of results will be divided into four subsections. The first two sections discuss the comparison between the CAA and experimental results. The third section discusses the transient behaviour of the acoustical propagation while the final section discusses the overall acoustical behaviour and distribution within the A-pillar region.

5.5.1 Comparison between Hybrid SWIFT CAA & Experimental Results – Cp RMS Pressure

Hybrid SWIFT CAA results of $C_p$ RMS distribution showed good correlation when compared to results obtained by Alam (2000). The comparison of results can be seen in Figures 5.2, 5.3, 5.7, 5.8, 5.11, 5.12, 5.16, 5.17, 5.18, 5.21, 5.22 and 5.23. The results were consistent with the findings of Murad (2007). The results of $C_p$ RMS distribution at various inlet velocities (60, 100 and 140 km/h) collapsed into a single curve for each model. As per Alam (2000), the modelled results showed that the airflow at various inlet velocities is Reynolds number independent.
Figures 5.60, 5.61 and 5.62 showed the Reynolds number sensitivity of peak \( C_p \) RMS. For all yaw angles, the variations of peak \( C_p \) RMS were small and the results obtained from the CAA simulation correlates well with the results obtained from Alam (2000). The maximum variation in CAA results obtained from circular models at 0° yaw was 0.050 as compared to 0.025, obtained experimentally. At -15° yaw, CAA results produced maximum variations of about 0.025, compared to 0.050, obtained experimentally. Finally, at +15° yaw, maximum variations obtained from both CAA and empirical results were 0.020. The maximum variation obtained from models with sharp edge models at 0° yaw was 0.20 as compared to 0.28, obtained experimentally. At -15° yaw, CAA results produced maximum variations of about 0.15, compared to 0.42, obtained experimentally. Finally, at +15° yaw, CAA results produced maximum variations of about 0.07, compared to 0.12, obtained experimentally. Discrepancies in the CAA results is caused by the discrepancies obtained in the earlier CFD models (discussed in Chapter 4) and also due to numerical discrepancies caused by the acoustical model used in software to determine the acoustical propagation estimation. This discrepancy in results is small and acceptable due to the constant fluctuating of acoustical pressure prior conversion to \( C_p \) RMS values.
Figure 5.61: Comparison between Numerical and Experimental Results of Peak Cp RMS for Reynolds Number Sensitivity, -15° Yaw

Figure 5.62: Comparison between Numerical and Experimental Results of Peak Cp RMS for Reynolds Number Sensitivity, +15° Yaw
5.5.2 Comparison between Hybrid SWIFT CAA & Experimental Results – PSD Distribution

Comparison of PSD distribution between results obtained from Hybrid SWIFT CAA and the experimental of Alam (2000) showed that good correlation with each other. A summary of PSD peak values obtained from the simulation, which is shown in Table 5.1.

Table 5.1: PSD and Frequency Peak for CAA

<table>
<thead>
<tr>
<th>Vehicle Models</th>
<th>0° (dB)</th>
<th>-15° (dB)</th>
<th>+15° (dB)</th>
<th>Peak Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>SE</td>
<td>100.0</td>
<td>99.0</td>
<td>98.0</td>
<td>100</td>
</tr>
<tr>
<td>Semi</td>
<td>103.0</td>
<td>102.0</td>
<td>98.0</td>
<td>100</td>
</tr>
<tr>
<td>LE</td>
<td>105.0</td>
<td>102.0</td>
<td>99.0</td>
<td>100</td>
</tr>
<tr>
<td>RE</td>
<td>117.0</td>
<td>116.0</td>
<td>109.0</td>
<td>250</td>
</tr>
<tr>
<td>SL</td>
<td>112.5</td>
<td>102.5</td>
<td>109.0</td>
<td>250</td>
</tr>
</tbody>
</table>

It can be seen from the Table that at 0° yaw, the peak PSD behind the A-pillar region was in the range of between 100-117-dB. When yawed, the peak PSD at the leeward region (-15°) was measured between 99-116-dB and the windward region (+15°) was measured between 99-109-dB. Overall, the measurement of peak PSD for the models was obtained between 99-117-dB. These findings were comparable to the results obtained from the works of other literatures. A study made by Hamel et al. (1996) showed that wind noise measurement behind the A-pillar region was measured at 110-dB. Lokhande et al. (2003) conducted an LES simulation of a generic side view mirror obtained OASPL peak of around 130-dB behind the mirror region. Fukushima et al. (1995) conducted a study on aerodynamic noise and found out that pressure fluctuation level generated behind the A-pillar region was around 90-100-dB. Stapleford et al. (1970) conducted experimental studies of aerodynamic noise generation and obtained an aerodynamic noise measurement at around 120-dB behind the A-pillar region.
Haruna et al. (1990) experimented using a production car in a wind tunnel, where the OASPL level measured at around 110-dB at 0° yaw. Haruna et al. (1992) developed a numerical model to estimate the distribution of surface pressure fluctuation for aerodynamic noise prediction and found out that the highest contributor of aerodynamic noise is the A-pillar with overall sound pressure level ranging between 110-130-dB. Haruna et al. (1992) developed a numerical model to analyze aerodynamic noise to mimic an A-pillar flow and found out that the highest peak of predicted OASPL was obtained at 110-dB. Bergamini et al. (1997) conducted both experimental and numerical simulation on an A-pillar bluff body and obtained sound pressure level of 115-dB from experimental measurement and 135-dB from the numerical model. Strumolo et al. (1998) simulated external aerodynamic noise to mimic the A-pillar region using CFD and obtained an OASPL of 125-dB from CFD and 120-dB empirically. Kumarasamy et al. (1999) conducted experimental and numerical simulation of A-pillar to predict aerodynamic noise and obtained an OASPL of 100-dB from experimental and 130-dB from CFD simulation.

On average the peak OASPL obtained from the works conducted from the literature was between 90-130-dB. The results obtained from Hybrid SWIFT CAA was between 99-117-dB, which makes it comparable with existing literatures. Discrepancies observed in the CAA results is caused by the discrepancies earlier obtained in the CFD models (discussed in Chapter 4) and also due to numerical error caused by the acoustical model used in software to determine the acoustical propagation estimation.

From Table 5.1, frequency corresponding to the peak PSD was also obtained. It can be seen that at peak PSD, the corresponding frequency was measured between 100-Hz to 250-Hz. According to Stapleford et al. (1970), Haruna et al. (1990) and Hanaoka et al. (1993), the highest sound pressure level obtained in region of low frequency was caused by large-scale turbulent structures at the area of flow separation. This was observed from the previous chapter especially when vehicle was exposed to a certain yaw angle. At low Mach number, this large-scale
turbulent structure changes the mean flow-field to produce low frequency pressure fluctuation, Simpson (1987, 1989). George (1990) and Callister et al. (1998) described this type of noise as broadband noise.

On average the dominant frequency level corresponding to the peak OASPL obtained from the works conducted from the literature was between 100 to 500-Hz. The results obtained from Hybrid SWIFT CAA was between 100 to 250-dB, which makes it comparable with existing literatures. As with previous results, discrepancies observed in the CAA results is caused by discrepancies obtained in the CFD models (discussed in Chapter 4) and also due to numerical error caused by the acoustical model used in software to determine the acoustical propagation estimation. George (1995) stated that aerodynamic noise is dominant in frequency region of between 500-Hz to 12,000-Hz. Buchheim et al. (1982) states A-pillar noise corresponds to region of low frequency between 250-Hz to 500-Hz. According to Haruna (1992) et al. the A-pillar noise corresponds to region of low frequency between 100 to 500-Hz. A study made by Hamel et al. (1996), have showed that the A-pillar region dominates the frequency spectrum range of lower than 1000-Hz. Li et al. (2003) conducted experimental and numerical study on reducing external aerodynamic noise and found out that peak sound pressure level was measure at frequency region of between 250 to 500-Hz using experimental method and between 500 to 1300-Hz using finite element method.

Table 5.2: PSD Peak and Overall Discrepancy between CAA and experimental

<table>
<thead>
<tr>
<th>Vehicle Models</th>
<th>Peak Discrepancy (dB)</th>
<th>Overall Discrepancy (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0°</td>
<td>-15°</td>
</tr>
<tr>
<td>SE</td>
<td>1.0</td>
<td>14.0</td>
</tr>
<tr>
<td>Semi</td>
<td>14.0</td>
<td>18.0</td>
</tr>
<tr>
<td>LE</td>
<td>5.0</td>
<td>13.0</td>
</tr>
<tr>
<td>RE</td>
<td>7.0</td>
<td>10.5</td>
</tr>
<tr>
<td>SL</td>
<td>10.5</td>
<td>26.5</td>
</tr>
</tbody>
</table>
The findings obtained using Hybrid SWIFT CAA (Figures 5.4, 5.5, 5.9, 5.13, 5.14, 5.19 and 5.24) showed that the PSD distributions for the models compared to the experimental results were under predicted. The average discrepancy between Hybrid SWIFT CAA and the experimental results were obtained and collated in Table 5.2. Average discrepancy obtained for the models was measured to be between 8.5-dB to 12.5-dB. Discrepancies observed in the CAA results is caused by the discrepancies earlier obtained in the CFD models (discussed in Chapter 4) and also due to numerical error caused by the acoustical model used in software to determine the acoustical propagation estimation.

Discrepancy of overall mean PSD value was comparable to the results obtained from the works of Lokhande et al. (2003), Ogawa et al. (1999), Uchida et al. (1999), which showed overall mean PSD discrepancies of around 15.0-dB for CAA modeling behind a bluff body region. Haruna et al. (1992) PSD analysis against measured results for A-pillar modeling showed that the numerical results were under predicted by about 20.0-dB, which was caused by insufficient amount of grid generation. Bergamini et al. (1997) experimental and numerical simulation in predicting aerodynamic noise of an A-pillar bluff body to predict vehicle aerodynamic noise resulted an over prediction by 20.0-dB. Strumolo et al. (1998) simulated external aerodynamic noise to mimic the A-pillar region using CFD with an over prediction of results by about 5.0-dB. Kumarasamy et al. (1999) experimental and numerical simulation of A-pillar to predict aerodynamic noise showed an over prediction of peak PSD by 30.0-dB. Overall PSD showed an under prediction by 10.0-dB. Li et al. (2003) conducted experimental and numerical study on reducing external aerodynamic noise on an actual model of a vehicle A-pillar with results showing a 5.0-dB to 10.0-dB over prediction.

On average the over/under prediction of PSD results obtained from the works conducted from the literature was between 5.0-dB to 30.0-dB. The results obtained from Hybrid SWIFT CAA showed an under prediction of between 8.5-dB to 12.5-dB, which makes it comparable with existing literatures.
5.5.3 Aero-acoustics Behaviour during Transient Conditions

The transient progression for each investigated scale model can be seen in Table 5.3. It can be seen based on the trend from the Table that the circular models investigated will reach a faster transition (EP/ST) and steady state (ET/SS) condition with increasing model windshield radii. This was observed when models were yawed at both 0° and 15° yaw respectively. For sharp edge models, faster transition and steady state were reached by the RE model as compared to the SL model. Average time taken for models to reach transition and steady state condition at 0° yaw was 0.06 and 0.21 seconds respectively. At 15° yaw, it took the models an average time of 0.07 and 0.40 seconds to reach transition and steady state condition respectively. It can be seen from the results obtained for the aero-acoustic transient time march behaviour that both transition and steady state condition were reached faster for models which does not experienced much turbulence activities behind the A-pillar region and at un-yawed condition where turbulent flow separation is less severe.

Table 5.3: Transient Progression of Aero-acoustics behind A-pillar Region

<table>
<thead>
<tr>
<th>Vehicle Models</th>
<th>EP/ST 0° (second)</th>
<th>EP/ST 15° (second)</th>
<th>ET/SS 0° (second)</th>
<th>ET/SS 15° (second)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SE</td>
<td>0.08</td>
<td>0.10</td>
<td>0.19</td>
<td>0.58</td>
</tr>
<tr>
<td>Semi</td>
<td>0.07</td>
<td>0.06</td>
<td>0.16</td>
<td>0.35</td>
</tr>
<tr>
<td>LE</td>
<td>0.02</td>
<td>0.04</td>
<td>0.13</td>
<td>0.26</td>
</tr>
<tr>
<td>RE</td>
<td>0.05</td>
<td>0.06</td>
<td>0.23</td>
<td>0.29</td>
</tr>
<tr>
<td>SL</td>
<td>0.09</td>
<td>0.09</td>
<td>0.34</td>
<td>0.50</td>
</tr>
<tr>
<td>Mean</td>
<td>0.06</td>
<td>0.07</td>
<td>0.21</td>
<td>0.40</td>
</tr>
</tbody>
</table>

Results from previous section have shown that for circular models, the LE model experienced less turbulence activity behind its A-pillar region compared to the SE model. This was due to the bigger windshield radius possessed by the LE model encouraging attached flow with minimum turbulent boundary layer generation throughout the A-pillar region downstream to the flow. Similar results obtained from the previous section between the sharp edge models showing that due to the
slant angle configuration of the A-pillar, turbulence separation and vortex
generation behind the A-pillar is more prominent for the SL model causing the
flow needing a longer time before it reaches a much more stable quasi-steady state
condition.

Results from Table 5.3 also showed that the aero-acoustic time march propagation
behaviour is dependent on the type of flow separation or vortex generation that is
occurring behind the A-pillar region. At 0° yaw, it can be seen that the SE and SL
models reaches steady state at a similar time. However, aero-acoustic time march
propagation trend in Figures 5.26 to 5.35 shows that for the sharp edge model, the
time march propagation trend is occurring at a much higher Cp RMS level, which
indicate a different level of aero-acoustic activity experienced by the model
behind the A-pillar region. Flow separation and the type of vortex generated for
the circular and the sharp edge models at 0° yaw was different, which was shown
and discussed in the previous chapter. For the circular models at 0° yaw, type of
flow separation and vortex generated are from two distinct sources, namely the
base A-pillar junction and the A-pillar apex/roof junction respectively. These flow
separation and vortex generation are much weaker than the ones produced by the
sharp edge models at 0° yaw, which is three dimensional helical cone shaped
vortex and are much stronger in turbulence intensity. The time march aero-
acoustic propagation at 15° yaw between the circular and sharp edge model are
much more consistent and predictable since both experienced a similar type of
flow separation and vortex generation behind the A-pillar region due to the
yawing orientation. It can be seen that the SL model takes longer to reach steady
state condition as compared to the LE model due to the severity of the turbulence
activity experience behind its A-pillar region downstream to the flow.

It can be said that the determining factor for aero-acoustic time-march propagation
behaviour for the circular and sharp edge model are the geometrical configuration
of the A-pillar slant and windshield radii. This will then shape the intensity of the
flow separation and turbulence vortex generation occurring behind the A-pillar
region downstream to the flow. Although flow will reach a much stable steady
state condition after certain time-march propagation, a certain level of unsteadiness is still present and that the vortex strength continues to change with time, Bearman et al. (1989).

5.5.4 CAA Behaviour Behind the A-pillar Region

The CAA behavior behind the A-pillar region was obtained from the Hybrid SWIFT CAA. The transient and steady state aero-acoustic propagation generated by flow separation and turbulence vortex development behind the A-pillar region was investigated. Only the CAA behavior of the circular models was assessed in this project.

Table 5.4 summarizes the OASPL generation on the vehicle surface during the initial transient propagation while Table 5.6 summarizes the OASPL generation on the vehicle surface after steady state conditions were reached. Table 5.5 summarizes the OASPL increment experienced by the models due to transient propagation to steady state condition. OASPL reduction as aero-acoustic propagation spreads away from the vehicle surface after steady state conditions were reached was summarized in Table 5.7. Finally, the OASPL propagation generated exclusively by the vortex on the vehicle surface during steady state conditions and away from the surface was summarized in Table 5.8 and 5.9 respectively.

Table 5.4: OASPL of Vehicle Surface during Initial Transient State

<table>
<thead>
<tr>
<th>Initial Surface Transient</th>
<th>Vehicle Models</th>
<th>0° (dB)</th>
<th>-15° (dB)</th>
<th>+15° (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SE</td>
<td>119.5</td>
<td>121.5</td>
<td>119.0</td>
</tr>
<tr>
<td></td>
<td>Semi</td>
<td>117.2</td>
<td>120.5</td>
<td>118.9</td>
</tr>
<tr>
<td></td>
<td>LE</td>
<td>115.8</td>
<td>120.0</td>
<td>118.7</td>
</tr>
<tr>
<td></td>
<td>Mean Average</td>
<td>117.5</td>
<td>120.7</td>
<td>118.9</td>
</tr>
</tbody>
</table>
Table 5.5: OASPL Increase from Transient to Steady on Vehicle Surface

<table>
<thead>
<tr>
<th>Vehicle Models</th>
<th>0° (dB)</th>
<th>-15° (dB)</th>
<th>+15° (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SE</td>
<td>3.2</td>
<td>1.5</td>
<td>3.5</td>
</tr>
<tr>
<td>Semi</td>
<td>4.3</td>
<td>2.2</td>
<td>2.8</td>
</tr>
<tr>
<td>LE</td>
<td>5.0</td>
<td>2.7</td>
<td>2.2</td>
</tr>
<tr>
<td>Mean Average</td>
<td>4.2</td>
<td>3.0</td>
<td>2.8</td>
</tr>
</tbody>
</table>

Table 5.6: OASPL on Vehicle Surface during Steady State

<table>
<thead>
<tr>
<th>Surface Steady</th>
<th>0° (dB)</th>
<th>-15° (dB)</th>
<th>+15° (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SE</td>
<td>122.7</td>
<td>123.0</td>
<td>122.5</td>
</tr>
<tr>
<td>Semi</td>
<td>121.5</td>
<td>122.7</td>
<td>121.7</td>
</tr>
<tr>
<td>LE</td>
<td>120.8</td>
<td>122.3</td>
<td>120.9</td>
</tr>
<tr>
<td>Mean Average</td>
<td>121.7</td>
<td>122.7</td>
<td>121.7</td>
</tr>
</tbody>
</table>

It can be seen from Table 5.4 and 5.6 that for the circular models, the OASPL range were measured at between 115-121-dB during the initial transient state and between 120-123-dB after it has reached steady state condition. On average the OASPL obtained from the existing literatures are between 90-130-dB, which makes the results obtained from Hybrid SWIFT CAA comparable with existing literatures, Hamel et al. (1996), Lokhande et al. (2003), Fukushima et al. (1995), Stapleford et al. (1970), Haruna et al. (1990), Haruna et al. (1992), Bergamini et al. (1997), Strumolo et al. (1998), Kumarasamy et al. (1999).

Mean average obtained for the models during initial transient propagation and after reaching steady state shows that OASPL magnitude are higher on the vehicle surface during yaw conditions. There is a 3-dB difference during initial transient propagation and a 1-dB difference during steady state condition when comparing vehicles under yawed and un-yawed scenario. This finding is in agreement with Haruna et al. (1990), in which stated that a difference up to 10-dB is present when
comparing OASPL magnitude between yawed and un-yawed vehicle. This finding was also in agreement with Fricke (1968 and 1971) in which stated that wall pressure fluctuations due to separated flow (which are more prominent at yaw angles, discussed in the previous chapter) are higher than those beneath a turbulent boundary layer (which are more prominent when vehicles are un-yawed, as discussed in the previous chapter). This high level of flow separation at yaw angle generates aerodynamic noise dipole effect, which is created due to the unsteady high pressure force fluctuation impingement on the wall surface, Hanaoka et al. (1993) and Callister et al. (1998).

It can be seen from Tables 5.4 and 5.6 as the windshield radii became larger, the OASPL magnitude at 0° yaw decreases by 3.7-dB during initial transient condition and 1.9-dB during steady state condition. At 15° yaw, the leeward region experienced a decrease in OASPL magnitude as the windshield radii becomes larger (1.5-dB reduction during initial transient condition and 0.7-dB reduction during steady state condition). In the windward region, the OASPL magnitude also experienced a decrease as the windshield radii become bigger (0.3-dB reduction during initial transient condition and 1.6-dB reduction during steady state condition). Overall, it can be seen from this finding that as the OASPL magnitude decreases with increases windshield radii. This finding is in agreement with Dobrzynski et al. (1994), Callister et al. (1998) and Alam (2000).

From Table 5.5, the models at 0° yaw experienced the largest increase in OASPL magnitude (4.2-dB) as it propagates from the initial transient condition to steady state condition. This is followed by the leeward (3.0-dB) and the windward (2.8-dB) region respectively when the vehicles are positioned to 15° yaw angle. As the windshield radii increases, the increase in OASPL magnitude as it propagates from initial transient condition to steady states increases as well.
Table 5.7: OASPL Reductions between Vehicle Surface and Domain during Steady State

<table>
<thead>
<tr>
<th>Reduction between Surface and Domain for Steady State</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vehicle Models</td>
</tr>
<tr>
<td>SE</td>
</tr>
<tr>
<td>Semi</td>
</tr>
<tr>
<td>LE</td>
</tr>
<tr>
<td>Mean Average</td>
</tr>
</tbody>
</table>

The reduction in OASPL magnitude as the aero-acoustics propagation moves away from the vehicle surface after reaching steady state condition is summarised in Table 5.7. Mean average experienced by the models at 15° yaw is higher (3.7-dB) as compared to when the models are not yawed (3.2-dB) due to a higher turbulence generation that consequently leads to a higher aerodynamic noise generation. This finding is in agreement with the findings of Stapleford et al. (1970) and George (1990), which states that depending on the turbulent intensity caused by flow separation, the aerodynamic noise generation on the vehicle surface are able to constitute an increase of about 17-dB to 20-dB when compared to background ambient noise. This finding is also in agreement with the finding of Nienaltowska (1993), in which Nienaltowska found that aerodynamic noise generation decreases as it moves away from the vehicle surface.

Table 5.8: OASPL of Vortex Propagation on Vehicle Surface at Steady State Condition

<table>
<thead>
<tr>
<th>Vehicle Models</th>
<th>Base Junction at Steady (dB)</th>
<th>A-pillar and Roof at Steady (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0°</td>
<td>-15°</td>
</tr>
<tr>
<td>SE</td>
<td>123.0</td>
<td>125.0</td>
</tr>
<tr>
<td>Semi</td>
<td>122.5</td>
<td>123.0</td>
</tr>
<tr>
<td>LE</td>
<td>120.0</td>
<td>121.0</td>
</tr>
<tr>
<td>Mean Ave.</td>
<td>121.8</td>
<td>123.0</td>
</tr>
<tr>
<td>Overall Mean On Surface</td>
<td>121.7</td>
<td>122.7</td>
</tr>
</tbody>
</table>

The OASPL magnitude at areas that were identified as source of flow separation and vortex generation was identified and summarised in Table 5.8. It can be seen
from the Table that the average OASPL magnitude is higher at 15° yaw as compared to 0° yaw. It can also be seen that the average OASPL magnitude is much higher at the A-pillar base junction as compared to A-pillar and roof junction for the circular models. In addition, it can be seen that the OASPL magnitude on areas of vortex propagation decreases as the windshield radii of the models increases for both at 0° and 15° yaw. There is a 4.3-dB difference between OASPL magnitude at the A-pillar base junction as compared to the A-pillar apex and roof junction at 0° yaw. At 15° yaw, the difference between the OASPL magnitude at the A-pillar base junction as compared to the A-pillar apex and roof junction is much smaller at 0.5-dB. Overall, mean surface OASPL magnitude behind the A-pillar region combined (A-pillar Base Junction, A-pillar apex and Roof) are higher compared to the overall mean OASPL magnitude on the surface of the vehicle. This is in agreement with the findings of Stapleford et al. (1970) and Strumolo et al. (1998).

It can be seen from Table 5.8 that the average OASPL magnitude on the A-pillar base junction is slightly higher compared to the overall OASPL mean on the surface of the models. At the A-pillar apex and roof junction, the overall OASPL mean on the surface is higher at 0° yaw but smaller at 15° yaw.

Table 5.9: OASPL Reductions between Vehicle Surface and Domain during Steady State at Vortex propagation area

<table>
<thead>
<tr>
<th>Vehicle Models</th>
<th>Base Junction/A-pillar/Roof Reduction at Steady (dB)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0°</td>
</tr>
<tr>
<td>SE</td>
<td>4.0</td>
</tr>
<tr>
<td>Semi</td>
<td>2.0</td>
</tr>
<tr>
<td>LE</td>
<td>1.0</td>
</tr>
<tr>
<td>Mean Average</td>
<td>2.3</td>
</tr>
</tbody>
</table>

The reduction in OASPL magnitude at the vortex propagation area as the aero-acoustics propagation moves away from the vehicle surface after reaching steady
state condition is summarised in Table 5.9. Results show an average of between 1.0-dB to 4.0-dB increase comparing OASPL values between surface and CAA domain. A higher surface and CAA domain difference is detected when the vehicle is yawed (3.0-dB mean average difference as compared to 2.3-dB when not yawed) as aerodynamic generation is higher. This finding is in agreement with the findings of Stapleford et al. (1970) and George (1990), which states that depending on the turbulent intensity caused by flow separation, the aerodynamic noise generation on the vehicle surface are able to constitutes an increase to about 17-dB to 20-dB when compared to background ambient noise. This finding is also in agreement with the finding of Nienaltowska (1993), in which Nienaltowska found that aerodynamic noise generation decreases as it moves away from the vehicle surface.
Chapter Six

CONCLUSIONS & RECOMMENDATIONS

6.1 Conclusions from Chapter 4

- The development of the CFD and CAA model consists of a four-stage process, in which, the first step was to first investigate and select the best grids configurations for both the circular and sharp edge A-pillar models at various respective yaw angles. The second and third stage process was to select the best turbulence and near wall model for the final CFD model. Turbulence and near wall models selected for comparison were taken from different commercial CFD software’s FLUENT and SWIFT AVL. The final stage in the development of the numerical model was to develop a CAA model for the aero-acoustics modelling.

- The grid configuration selected for this project was polyhedral grids from SWIFT AVL. The total element count after final refinement was within an acceptable limit of less than two million grids.

- For the circular models (SE, Semi and LE) at 0°, the final turbulence and NWM chosen for the project was the standard $k – \varepsilon$ turbulence model with the NWM of Chieng and Launder (1980).

- For the circular models at 5°, 10° and 15° yaw angles, together with the sharp edge models at 0°, 5°, 10° and 15° yaw, the final turbulence and NWM model chosen for the project was the RSM with the WEB NWM of Manceau and Hanjalic (2002).

- Validation of the final CFD model against the experimental data of Alam (2000) resulted in good correlations with mean error deviation obtained within the acceptable recommended value of 20%.

- The CAA model developed for this project resulted in a total grid count of around 35,000 grids for the selected CAA domain. Final CAA modelling was conducted using SWIFT CAA and assessment for aero-acoustics
behaviour behind the A-pillar region was conducted only for the circular models due to their better correlations with experimental data.

- Comparison of mean Cp values between the CFD and experimental results in this chapter have shown that error of deviation for the circular models (SE, Semi and LE) have all fallen within the recommended 20% margin. The best performing model was the SE model at 0° which yielded an under prediction of 8.2%. The worse performing model was the LE model at 15° yaw which yielded an under prediction of 24.5%. For sharp edge models (RE and SL) error of deviation for models at yaw angles of 0°, 5°, 10° and 15° yaw has also fallen within the recommended 20% margin. The best performing model was the RE model at 0° which yielded an under prediction of 11.9%. The worse performing model was the SL model at 0° yaw which yielded an under prediction of 29.8%.

- The size of the vortex measured for the various models at different yaw angles in the leeward region of the flow suggest that with the circular models (SE, Semi and LE) and the sharp edge models (RE and SL), the vortex size increases with yaw angles.

- Vortex for the circular models at 40% scale was measured between 115mm to 340mm in the horizontal component and between 102mm to 430mm in the vertical component. Vortex measurement for the circular models at 40% scale shows that as the yaw angle increases from 0° to 15° yaw, the vortex experiences a size increment of 225 mm in the horizontal component and 328 mm in the vertical component.

- Vortex for the RE model at 40% scale was measured between 80mm to 450mm in the horizontal component and between 200mm to 590mm in the vertical component. Vortex measurement for the RE model at 40% scale shows that as the yaw angle increases from 0° to 15° yaw, the vortex experiences a size increment of 370 mm in the horizontal component and 390 mm in the vertical component.

- Vortex for the SL model at 40% scale was measured between 110mm to 440mm in the horizontal component and between 360mm to 380mm in the vertical component. Vortex measurement for the SL model at 40% scale
shows that as the yaw angle increases from 0° to 15° yaw, the vortex experiences a size increment of 330 mm in the horizontal component and 60 mm in the vertical component.

- The results obtained from the CFD analysis shows that for both the circular and sharp edge models, the source of vortex separation behind the A-pillar region originated from the junction of the A-pillar base, the A-pillar apex and the front side window and roof junction. The mechanism of flow separation for both the circular and sharp edge models was due to trailing edge separation.

- For circular models, the shape of the vortices that takes place at 0° yaw took a physical form of a two-dimensional quasi-elongated oval. At 15° yaw, the shape of the vortices for the circular models took an overall physical form of a three-dimensional mixture of a quasi circular and cone shaped helical vortex, which was a chaotic combination of two-dimensional and three dimensional vortices originating from the A-pillar base junction, apex and roof junction.

- For the RE sharp edge model, shape of the vortex that takes place at 0° yaw took a physical form a mixture of two-dimensional free trailing vortex originated from the base of the A-pillar junction, while vortex generated from the A-pillar apex formed a mixture of three-dimensional quasi circular and cone shape helical vortex. The vortex generated from the roof junction formed a three-dimensional circular shape vortex. All three form of vortices flowed in stream wise direction, downstream to the flow. For the RE shape edge model at 15° yaw, the shape of the vortex that takes place at the A-pillar base junction and A-pillar apex was similar to that at 0° yaw.

- For the SL sharp edge model, shape of the vortex that takes place at 0° yaw took a physical form of a three-dimensional vertically elongated cone shape helical vortex propagating downstream to the flow. At 0° yaw and 15° yaw, the shape of the vortex at the A-pillar base junction, the A-pillar slant edge and the roof junction combine to form the three dimensional
conical helical vortex, making the turbulent intensity stronger as the yaw angle increases.

- Observation of various shapes of windshield radii and slant angle models simulated in this project showed that the various geometrical configurations of the windshield radii and slant angle governs the behaviour pattern of vortex generation behind the A-pillar region when exposed to yawed or un-yawed position. The observations have showed that the vortex generated behind the A-pillar region was lower in intensity and size in the circular models as compared to the sharp edge models. The circular shape models windshield geometrical configuration and slant angle created three different vortex formations, which shape the direction of flow. The vortex generated from the various sources created a scenario whereby the airflow path interferes with each other, creating a chaotic quasi-circular vortex structure that prevents the formation of highly intense vortices behind the A-pillar region. The sharp edge models windshield geometrical configuration and slant angle created a scenario whereby the airflow path from various directions converges together effectively to form a highly intense three-dimensional helical cone shape vortex all the way downstream to the flow. This produces region of high turbulent intensity behind the A-pillar region. When subjected to yaw angles, the airflow change angle of attack causes early turbulent boundary layer transition, which will increase the vortex magnitude, and intensity further, which was shown in the increase in the formation and size of the vortex.
6.2 Conclusions from Chapter 5

- Hybrid SWIFT CAA results of $C_p$ RMS distribution showed good correlation when compared to results obtained by Alam (2000). The results of $C_p$ RMS distribution at various inlet velocities (60, 100 and 140 km/h) showed good correlation with the existing experimental results of Alam (2000).

- On average the peak OASPL obtained from the works conducted from the literature was between 90-130-dB. The results obtained from Hybrid SWIFT CAA was between 99-117-dB, which makes it comparable with existing literatures. The average discrepancy between Hybrid SWIFT CAA and the experimental results for PSD distributions was measured to be between 8.5-dB to 12.5-dB. On average the over/under prediction of PSD results obtained from the works conducted from the literature was between 5.0-dB to 30.0-dB, which makes it comparable with existing literatures.

- On average the dominant frequency level corresponding to the peak OASPL obtained from the works conducted from the literature was between 100 to 500-Hz. The results obtained from Hybrid SWIFT CAA was between 100 to 250-dB, which makes it comparable with existing literatures. The highest sound pressure level obtained in region of low frequency at low Mach number was caused by large-scale turbulent structures at the area of flow separation. The large-scale turbulent structure changes the mean flow-field to produce low frequency pressure fluctuation, which is often described as broadband noise.

- The transient progression for each investigated scale model shows that the circular models investigated will reach a faster transition (EP/ST) and steady state (ET/SS) condition with increasing windshield radii and also when not exposed to yaw condition. This is because as models windshield radii increases, aerodynamic flow behind the A-pillar region experiences less turbulence activities. Similar explanation is valid for models that are not yawed. For sharp edge models, faster transition and steady state were
reached by the RE model as compared to the SL model. Turbulence separation and vortex generation behind the A-pillar is more prominent for the SL model causing the flow needing a longer time before it reaches a much more stable quasi-steady state condition. Average time taken for models to reach transition and steady state condition at $0^\circ$ yaw was 0.07 and 0.40 seconds respectively. At $15^\circ$ yaw, it took the models an average time of 0.06 and 0.21 seconds to reach transition and steady state condition respectively. Although flow will reach a much stable steady state condition after certain time-march propagation, a certain level of unsteadiness is still present and that the vortex strength continues to change with time.

- For the circular models, the OASPL range on the surface were measured at between 115-121-dB during the initial transient state and between 120-123-dB after it has reached steady state condition. On average the OASPL obtained from the existing literatures are between 90-130-dB, which makes the results obtained from Hybrid SWIFT CAA comparable with existing literatures.

- As models propagates from the initial transient condition to steady state condition, models at $0^\circ$ yaw experienced the largest increase in OASPL magnitude (4.2-dB). This is followed by the leeward (3.0-dB) and the windward (2.8-dB) region respectively when the vehicles are positioned to $15^\circ$ yaw.

- Results show that OASPL magnitude is higher on the vehicle surface during yaw conditions. There’s a 3-dB difference during initial transient propagation and a 1-dB difference during steady state condition when comparing vehicles under yawed and un-yawed scenario. This results is comparable with existing literatures which stated that a difference up to 10-dB is present when comparing OASPL magnitude between yawed and un-yawed vehicle.

- Results also show that OASPL magnitude on the vehicle surface decreases with increases windshield radii. This finding is in agreement with the existing literatures. As the windshield radii becomes bigger, the OASPL magnitude at $0^\circ$ yaw decreases by 3.7-dB during initial transient condition.
and 1.9-dB during steady state condition. At 15° yaw, the leeward region experienced a decrease in OASPL magnitude as the windshield radii becomes larger (1.5-dB reduction during initial transient condition and 0.7-dB reduction during steady state condition). In the windward region, the OASPL magnitude also experienced a decrease as the windshield radii become bigger (0.3-dB reduction during initial transient condition and 1.6-dB reduction during steady state condition).

- This findings shows that aerodynamic noise generation decreases as it moves away from the vehicle surface. The reduction in OASPL magnitude as the aero-acoustics propagation moves away from the vehicle surface after reaching steady state condition shows that the mean average experienced by the models at 15° yaw is higher (3.7-dB) as compared to when the models are not yawed (3.2-dB). This finding is in agreement with existing literatures, in which shows that the aerodynamic noise generation on the vehicle surface are able to constitutes an increase to about 17-dB to 20-dB when compared to background ambient noise.

- Overall, mean surface OASPL magnitude at the vortex source region (A-pillar Base Junction, A-pillar apex and Roof) is slightly higher compared to the overall mean OASPL magnitude on the surface of the vehicle. This is in agreement with the findings of existing literatures.

- There is a 4.3-dB difference between OASPL magnitude at the A-pillar base junction as compared to the A-pillar apex and roof junction at 0° yaw. At 15° yaw, the difference between the OASPL magnitude at the A-pillar base junction as compared to the A-pillar apex and roof junction is much smaller at 0.5-dB.

- The reduction in OASPL magnitude at the vortex propagation area as the aero-acoustics propagation moves away from the vehicle surface after reaching steady state condition shows an average of between 1.0-dB to 4.0-dB increase comparing OASPL values between surface and CAA domain. This finding is in agreement with the existing literatures, which states that depending on the turbulent intensity caused by flow separation, the aerodynamic noise generation on the vehicle surface are able to
constitutes an increase to about 17-dB to 20-dB when compared to background ambient noise.

This PhD project has achieved its aim and objectives. The PhD research project of Alam was extended. Using CFD approach the visualization of the vortex structure behind the A-pillar region was modeled and investigated. The size and structure of the vortex that was developed from various A-pillar windshield radii exposure at various yaw angles was investigated and quantified. This was not achieved by Alam using the experimental approach. The mechanism of A-pillar vortex generation, the transient and also acoustical behaviour was also investigated in this project and understood. The main achievement of this project is in the development of methodology in the understanding the aerodynamic and aero acoustic behaviour behind the A-pillar region of a vehicle. This study will be beneficial to the automotive industries in the reduction of airborne noise for passenger comfort. Further recommendation for future study is outline below.

### 6.3 Further Recommendations

Below are some of the recommendations for future work that were not included in this project. The recommendations are as follows:

- The aero acoustics pattern of behaviour and distribution on the model surface and the surrounding CAA domain was investigated in this project through the generation of OASPL. The scope of this project however serves only as a ground work study for the aero acoustics modelling within the vehicle A-pillar domain. However, it is recommended that further validation to be conducted which includes empirical validation of the model to identify proper mechanism of aero acoustics generation, i.e. monopole or dipole type generation. It is also recommended that further refinement can be conducted that includes grid independency testing for the CAA domain to determine the most optimum grids that are required for each CAA domain model. Apart from discrepancies caused by earlier CFD models (discussed in Chapter 4) and also due to numerical error
caused by the acoustical model used in software to determine the
coustical propagation estimation, lack of grid refinement of the CAA
domain might also contributed to the discrepancies caused.

The scope of this project was to use steady state RANS CFD results as
boundary conditions for the CAA modelling. Transient RANS CFD
modelling was not conducted due to time constraints in finishing this
project and it is recommended for future work in order to investigate and
analyse qualitatively the A-pillar vortex behaviour under transient
condition. In conducting transient modelling of the A-pillar aerodynamics,
it is recommended that LES modelling is used as it offers a higher level of
modelling accuracy without having to resort to extreme grid generation
capacity.

This scope of this project was limited to investigate and analyse generally
and qualitatively the pattern of behaviour of the A-pillar aerodynamics,
specifically the mechanism and formation of the A-pillar vortex at various
yaw angles and at various windshield radii. The behaviour within the
turbulence boundary layer separation was not carried out due to time
constraints and is still unknown. It is recommended for future works that
that a more detailed investigation be carried out on boundary layer
behaviour behind the A-pillar region in order to investigate the turbulence
unsteadiness and physics underneath the boundary layer separation. This
can be done by investigating the Reynolds stress behaviour within the
boundary layer flow in trying to find any correlation with acoustical
generation and propagation.
REFERENCES


Buley, M.D., “Industrial Noise Control”, Swinburne University of Technology Lecture Notes, School of Mechanical and Manufacturing Engineering, Swinburne University of Technology, Melbourne Australia, 1997.


Imperial College, “Lecture Notes on Turbulence”, Imperial College, U.K.


Swinburne University Lecture Notes in Computational Fluid Dynamics, Swinburne University of Technology, Melbourne, Australia, 2000.


BIBLIOGRAPHY

