NUMERICAL SIMULATIONS ON THE EFFECTS OF EDGE DETAILS ON AERODYNAMIC CHARACTERISTICS OF LONG SPAN BRIDGE DECK SECTIONS.

by

Richard A. Obisanya

Supervisor:

Prof. A.G. Robins

A Thesis Submitted for the degree of Doctor of Philosophy (PhD)

Faculty of Engineering and Physical Sciences
University of Surrey
May 2010
Numerical Simulations on the Effects of Edge Details on Aerodynamic Characteristics of Long Span Bridge Deck Section

Richard A. Obisanya
Numerical Simulations on the Effects of Edge Details on Aerodynamic Characteristics of Long Span Bridge Deck Section

Richard A. Obisanya

Abstract

The design of Long Span Bridges involves complex analysis of the interaction between fluid and bluff body (Fluid Structure Interaction). In the past, the aerodynamic characteristics needed for the design of long span bridge deck sections have been obtained via wind tunnel tests. Recent advances in turbulence modeling, computational fluid dynamics and the increasing affordability of computers have made numerical modeling of these complex studies possible.

Much research has been carried out on the applicability of CFD in the study of bluff body aerodynamics, and less relating to long span bridges. Unfortunately, due to computational costs and sometimes lack of complete details from the wind tunnel test results, these studies have been limited in scope; usually the work is 2-dimensional and often limited to the basic section without the parapets and equipment that are part of the super structure. Also, some experimental work has been done on shaped bluff body sections, such as rectangular cylinders, which has provided useful but limited application to a bridge deck section.

The work described in this theses consist of modeling and simulation of the sectional wind tunnel test of the Carquinez Strait bridge in California, a real long span bridge deck section. The modeling incorporates the often ignored but important details such as parapets, barriers and most importantly, the effects of the shape of different edge details on aerodynamic characteristics such as lift, drag and moment coefficients, as well as the
flow pattern created by the different edge details in the shedding of vortices in their wakes.

The simulations were carried out using the $k-\omega$ based Shear Stress Transport RANS (Reynolds Averaged Navier Stokes) turbulence model at an average wind velocity of 3.2 m/s with angles of attack of $\pm 10^\circ$. The basic deck section of the Carquinez Strait Bridge is of trapezoidal box girder with sharp edge detail type, this cross section was modified by modifying the edge detail and replacing it with three different types of details; a round edge detail, an oval and a triangular shaped edge type. Additional studies include the removal of the barrier, parapet and equipment to see their effect and the roles played by them in the aerodynamic static force response and the flow physics.

Two grid types were explored to determine the most accurate; tetrahedral and hexahedral dominated meshes. Next, determining the appropriate RANS turbulence model, from the matrix of grid and turbulence model emerges the numerical simulation.

Once the wind tunnel test results were corrected for errors, the results from numerical modeling compares very well with the static wind tunnel test, thereby validating the choice of turbulence model and grid type, and demonstrating the viability of CFD in long span bridge design.

The results of the fluid flow around the differently modified edge details shows how the mechanics of vortex induced vibration develops off of the recirculating air underside the exterior web at the trailing edge, because of the variation in the velocity of air in this region due to the different edge details, it is reasonable to make deductions on stability. In the simulations where the parapets, barriers and equipments are removed off of the deck sections, the response are markedly different, revealing that they are critical and as important as the edge detail chosen during the preliminary design.
Acknowledgement

I am most grateful to God for His grace and mercy on my life, my former colleagues in California at Office of Structures Design for providing me the details and the Wind Tunnel test results of several bridges for me to use, especially Tony Marquez, Project Manager on the Carquinez Strait Bridge.

I am thankful to the following for the reasons stated:

CFX Harwell- for providing the CFX software.

Paul and Fran Handrick: I wish the whole world is just like both of you, may God bless both of you greatly. Abraham and Sarah Oshuntola- Thank you good friends and for taking care of me when I was sick. Tai Oduyemi, Clement Bamgbade- for friendship and encouragement.

Paul Evans: Giving me your car was huge, may the God you serve bless and reward you greatly. Maria Quipildor: Gracias pastor para su paryers, usted cumplió sinceramente el carga de Cristo y yo soy endeudado a usted. Diane Ingebrigtsen: Gracias para su ayuda y oraciones.

Finally, I am grateful to Prof. Alan Robins for rescuing this work. I am so fortunate to know a man of such integrity.
# Contents

Abstract ........................................................................................... I

Acknowledgements .............................................................................. II

Contents .................................................................................................. III

List of Figures

List of Tables

List of Symbols and Acronyms

1 Introduction

1.0 Introduction.....................................................................................1

1.1 Scope of Work ..............................................................................6

1.2 Organization of the thesis.............................................................9

1.3 Summary.......................................................................................13

2 Literature Review

2.0 Introduction.....................................................................................14

2.1 Computational Wind Engineering ..............................................17

2.2 Flutter............................................................................................23

2.3 Other types of Aeroelastic Phenomena.......................................29

2.4 Vortex Induced Vibrations ..........................................................30

2.5 Vortex Induced Vibrations: The lock-in Phenomenon ...............47

2.6 Reynolds Number Effect ..............................................................49

2.7 Effect of Shapes on flows .............................................................56

2.8 Numerical Simulations .................................................................61

2.9 Summary.......................................................................................72
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.3</td>
<td>The Study Models</td>
<td>185</td>
</tr>
<tr>
<td>6.4</td>
<td>Flow Parameters</td>
<td>189</td>
</tr>
<tr>
<td>6.5</td>
<td>Numerical Modeling &amp; Boundary Conditions</td>
<td>192</td>
</tr>
<tr>
<td>6.6</td>
<td>Steady State and Transient Flows</td>
<td>202</td>
</tr>
<tr>
<td>6.6.1</td>
<td>Adopted Procedure</td>
<td>202</td>
</tr>
<tr>
<td>6.7</td>
<td>Parametric Study on Variation of Turbulence Intensity</td>
<td>207</td>
</tr>
<tr>
<td>6.8</td>
<td>Parametric Study on Variation of Turbulence Length Scale</td>
<td>208</td>
</tr>
<tr>
<td>6.9</td>
<td>Summary</td>
<td>209</td>
</tr>
<tr>
<td>7</td>
<td>Effect of Edge Details</td>
<td></td>
</tr>
<tr>
<td>7.0</td>
<td>Introduction</td>
<td>211</td>
</tr>
<tr>
<td>7.1</td>
<td>Error Analysis and Corrections on the Wind Tunnel Test Results</td>
<td>215</td>
</tr>
<tr>
<td>7.2</td>
<td>Comparison of Static Aerodynamic Characteristics</td>
<td>219</td>
</tr>
<tr>
<td>7.3</td>
<td>Mean Flow Distributions</td>
<td>253</td>
</tr>
<tr>
<td>7.4</td>
<td>Vortex shedding</td>
<td>262</td>
</tr>
<tr>
<td>7.5</td>
<td>Turbulence Kinetic Energy Distributions</td>
<td>272</td>
</tr>
<tr>
<td>7.6</td>
<td>Effect of Grid Types on Flow Distributions</td>
<td>282</td>
</tr>
<tr>
<td>7.7</td>
<td>Parametric Study on the Effect of Certain Boundary Conditions on Simulated Results</td>
<td>284</td>
</tr>
<tr>
<td>7.8</td>
<td>Summary</td>
<td>287</td>
</tr>
<tr>
<td>8</td>
<td>Conclusion and Proposals for future research</td>
<td></td>
</tr>
<tr>
<td>8.0</td>
<td>Introduction</td>
<td>288</td>
</tr>
<tr>
<td>8.1</td>
<td>Contributions &amp; Summary of the present study</td>
<td>289</td>
</tr>
<tr>
<td>8.2</td>
<td>Recommendations for Future research</td>
<td>292</td>
</tr>
<tr>
<td>8.3</td>
<td>CFD Modeling – Some Important Issues</td>
<td>294</td>
</tr>
</tbody>
</table>
List of Figures

Figure 1a Messina Strait Bridge Deck Section ......................................................... 4
Figure 1b Tsing Ma Bridge Deck Section ................................................................. 4
Figure 1c Normandie Bridge Deck Section ............................................................. 5
Figure 1d Little Belt Bridge Deck Section ............................................................... 5
Figure 1e Great Belt East Bridge ............................................................................ 5
Figure 2a Torsional Mode of the Tacoma Narrows Bridge .................................... 16
Figure 2b Failure of Tacoma Narrows Bridge ....................................................... 16
Figure 2.1a Sectional Degree of freedom for aerodynamic forces ....................... 19
Figure 2.1b F and G of the Theodorsen Circulation function ................................. 20
Figure 2.3a Deck model of the Great Belt East Bridge for Flutter ....................... 26
Figure 2.3b Wind induced Aeroelastic phenomenon ............................................ 28
Figure 2.4a Examples of Boundary Layers ............................................................. 32
Figure 2.4b Boundary layer of Flat Plate ............................................................... 33
Figure 2.4c Formation of Boundary Layer due to Viscosity .................................. 33
Figure 2.4d Vortex Shedding from a square cylinder near a wall ....................... 45
Figure 2.4e Growth of Boundary layer ................................................................. 46
Figure 2.4f Influence of Strouhal Number ............................................................. 46
Figure 2.4g Plots of Vorticity showing wake activity changes with boundary layer... 47
Figure 2.4h Reynolds number effect on drag coefficients ..................................... 35
Figure 2.4i Examples of Bluff body models .......................................................... 37
Figure 2.4j Strouhal number as function of L/D for various sections .................... 38
Figure 2.4k Strouhal number vs. base suction for semi circular and circular cylinders 39
Figure 2.4l-1 Laminar periodic wake at Re=140 ..................................................... 41
Figure 2.4l-2 Periodic wake at Re=300 ............................................................... 41
Figure 2.41-3  Laminar Synchronized wake an oscillating cylinder.........................42
Figure 2.4m  Vortices shed during half cycle for oscillating cylinder......................43
Figure 2.4n  Classes of Vortex formation from several bluff sections......................44
Figure 2.5a  Lock-in Phenomenon...........................................................................49
Figure 2.6a-1  Classical example for strong Reynolds number effect..........................51
Figure 2.6a-2  Flow regime around trapezoidal box girder........................................52
Figure 2.6a-3  Reynolds number effect on lift coefficient on box girder......................53
Figure 2.6a-4  Flow topologies around the Great Belt East Bridge............................55
Figure 2.7a  Experimental setup of flow between two cylinders...............................58
Figure 2.7b  Flow patterns and streamline representation of two cylinders................58
Figure 2.7c  Free end shapes tested for flow patterns..............................................60
Figure 2.7d  Spatial Distribution and Flow past shaped cylinders.............................61
Figure 2.8a  Vortex shedding past rectangular cylinders by DVM...............................64
Figure 2.8b  Geometry used in Galloping studies....................................................67
Figure 2.8c  Vorticity and pressure field due to galloping (rectangular section)...........67
Figure 2.8d  Vorticity and pressure field due to galloping (semi-circular section)........67
Figure 2.9  Geometry and Flow field of generic bridge............................................69
Figure 2.9b  Aki-nada Ohashi Bridge Deck Section..................................................70
Figure 3.4  Overproduction of turbulent kinetic energy by $k-\varepsilon$ model..................89
Figure 3.5a  Effect of grid types on the SST turbulence model..................................96
Figure 3.5b  Geometries of Models from the AIAA Drag prediction Workshop...........97
Figure 3.5c  Drag polar from the AIAA Drag prediction Workshop............................97
Figure 3.9  Computational cell................................................................................98
Figure 4.1a  Tetrahedral Element..............................................................................113
Figure 4.1b  Hexahedral Elements...........................................................................114
Pyramid Element
Unstructured Grid of Tetrahedra
Unstructured Grid Discretization
Octant Subdivision
Mesh control types
Convex Polygon & Voronoi diagram
Example of Delaunay criteria
Structured Mesh Discretization and organization of nodes
Multi block Structured grid
Mapping of boundaries between computational & physical domain
Mapping from computational space to domain space
Subdivision of near wall region in boundary layer
Turbulent Boundary Layer Distribution Near-Wall Region
Characteristic roughness height within viscous sub layer
Boundary layer separation
Inflated mesh at near wall regions
Undesirable skewness in grid cells
Grid Generation Process
Plane through tetrahedron dominated mesh
Plane through hexahedron dominated mesh
Cluster layout
Cluster nodes for parallel simulations
Partitioning process
Node based and Element based partitioning
Wedge Elements

Figure 4.1c
Figure 4.1d
Figure 4.3a
Figure 4.3b
Figure 4.3c
Figure 4.4
Figure 4.6
Figure 4.7
Figure 4.8a
Figure 4.8b
Figure 4.9a
Figure 4.10
Figure 4.10b
Figure 4.10c
Figure 4.11a
Figure 4.11b
Figure 4.12
Figure 4.13
Figure 4.14
Figure 4.15a
Figure 4.15b
Figure 5.1a
Figure 5.1b
Figure 5.4
Figure 5.5
Figure 7.3a Drag Coefficient of Tacoma Narrows Bridge ........................................... 226
Figure 7.3b Drag Coefficients New Burrard Inlet Crossing ........................................ 226
Figure 7.3c Cross Section of the New Burrard Inlet Crossing ..................................... 226
Figure 7.4 Drag Coefficients of Model w/o Parapets ................................................ 227
Figure 7.5a Lift Coefficients for Simulated Models with Parapets and Barrier Railings  .................................................................................................................. 231
Figure 7.5b Lift Coefficients-West Wind without Parapets ......................................... 232
Figure 7.6a Lift Force @ 0°: West Wind (With parapets) ........................................... 233
Figure 7.6b Lift Force @ 0°: West Wind (Without parapets) ....................................... 234
Figure 7.7a Comparison of Moment Coefficients-With Parapets present on Deck .... 237
Figure 7.7b Comparison of Moment Coefficients-w/o Parapets present on Deck ..... 238
Figure 7.8i Comparison of Pressure distribution on Deck sections ............................. 240
Figure 7.9a Contour & Velocity (m/s) distribution-Model 1(Center plane Y1) .......... 253
Figure 7.9b Contour & Velocity (m/s) distribution-Model 1(Plane Y3) ................. 254
Figure 7.9c Contour & Velocity Distribution-Model 2(Center plane Y1) .......... 255
Figure 7.9d Contour & Velocity (m/s) Distribution-Model 2(Plane Y3) .......... 256
Figure 7.9e Contour & Velocity (m/s) Distribution-Model 3(Plane Y1) .......... 257
Figure 7.9f Contour & Velocity (m/s) Distribution-Model 3(Plane Y3) .......... 258
Figure 7.9g Contour & Velocity (m/s) Distribution-Model 4(Center plane Y1) .... 259
Figure 7.9h Contour & Velocity (m/s) Distribution-Model 4(Plane Y3) .......... 260
Figure 7.10a Recirculation at Trailing Edge of Model 1 ........................................... 263
Figure 7.10b Recirculation at Trailing Edge of Model 2 ........................................... 264
Figure 7.10c Recirculation at Trailing Edge of Model 3 ........................................... 265
Figure 7.10d Recirculation at Trailing Edge of Model 4 ........................................... 267
Figure 7.10e Vortex Shedding in the wake of Model 1 @ α=0 .................................... 267
List of Tables

Table 4.1  Model properties & comparison of grid types ..........................................156
Table 4.2a Partitioning Information on the Hexahedron dominated model ..................158
Table 4.2b Partitioning Information on Tetrahedron dominated model .....................158
Table 4.3  Comparison of Tetrahedron, Hexahedron and wind tunnel test .................159
Table 4.4  Mesh Statistics for Parametric Studies on Domain Grid Resolution ..........161
Table 4.5  Comparison of Static Forces for Parametric Studies on Domain Grid
Resolution .........................................................................................................................161
Table 6.2  Mechanical Properties of the 050-HR LVDT Transducers ..........................179
Table 6.7  Variation of Turbulent Intensity and Static Force .....................................207
Table 6.8  Variation of Turbulent Length Scale and Static Forces ............................208
Table 7.2  Comparison of Structural Properties of Various Models ..........................219
Table 7.3a Comparison of Wind Tunnel Test results and Numerical Simulations for
Drag ..................................................................................................................................221
Table 7.3b Comparison of Wind Tunnel Test results and Numerical Simulations
for drag .............................................................................................................................222
Table 7.4  Comparison of Wind Tunnel test (West Wind) results and Numerical Simulation
..........................................................................................................................................229
Table 7.5  Comparison of Wind Tunnel test (West Wind) results and Numerical Simulation
..........................................................................................................................................235
Nomenclature

\[ A \] = Cross Sectional Area \((mm^2)\)

\[ A_n^* \] = Flutter derivative variable

\[ b \] = Semi Chord \((mm)\)

\[ B \] = Breadth of Deck Section \((mm)\)

\[ C(k) \] = Complex Theodorsen circulation function

\[ C_D \] = Drag Coefficient (dimensionless)

\[ C_{du} \] = Uncorrected drag coefficient

\[ C_L \] = Lift Coefficient (dimensionless)

\[ C_{lu} \] = Uncorrected lift coefficient

\[ C_M \] = Moment Coefficient (dimensionless)

\[ C_{mu} \] = Uncorrected moment coefficient

\[ C_p \] = \(k-e\) Turbulence Model constant

\[ CFL \] = Courant-Friedrichs-Levy number

\[ D \] = Depth (Deck Section) \((mm)\)

\[ D_k \] = Hydraulic diameter

\[ f \] = frequency

\[ F_D \] = Drag Force (N)

\[ F_L \] = Lift Force (N)

\[ H_n^* \] = Flutter derivative variable

\[ h \] = Bending or Heaving Deflection \((mm)\)

\[ h_{st} \] = Specific Static enthalpy

\[ h_s \] = Height of section

\[ \dot{h} \] = Heaving acceleration \(\left(\frac{mm}{s^2}\right)\)

\[ \ddot{h} \] = Heaving velocity \((mm/s)\)

\[ I \] = Intensity (Turbulence)

\[ I_n \text{ or } I \] = Mass moment of Inertia

\[ \text{xv} \]
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{mn}$</td>
<td>Moment of Inertia about $mn$ axis</td>
</tr>
<tr>
<td>$K_i$</td>
<td>Wind tunnel correction constant for solid blockage</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$M_v$</td>
<td>Model volume</td>
</tr>
<tr>
<td>$P_i$</td>
<td>Shear production of Turbulence</td>
</tr>
<tr>
<td>$p, p_{true}$</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>$p_{ref}$</td>
<td>Reference pressure</td>
</tr>
<tr>
<td>$p_{tot}$</td>
<td>Total Pressure</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius of gyration</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds Number</td>
</tr>
<tr>
<td>$S_E$</td>
<td>Energy source</td>
</tr>
<tr>
<td>$S_M$</td>
<td>Momentum source</td>
</tr>
<tr>
<td>$S_i, S_R$</td>
<td>Strouhal number</td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
</tr>
<tr>
<td>$u$</td>
<td>Fluctuating velocity component in turbulent flows.</td>
</tr>
<tr>
<td>U</td>
<td>Free stream velocity</td>
</tr>
<tr>
<td>$V_D$</td>
<td>Fluid Domain Volume</td>
</tr>
<tr>
<td>$V_f$</td>
<td>Critical flutter wind speed</td>
</tr>
<tr>
<td>$y'$</td>
<td>Dimensionless distance from the wall, used to check location of first node.</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Air Density</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Turbulence dissipation rate</td>
</tr>
<tr>
<td>$\varepsilon_{sh}$</td>
<td>Blockage correction</td>
</tr>
<tr>
<td>$\varepsilon_{wb}$</td>
<td>Wake blockage correction</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Wind correction parameter</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>Time step</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>Change in Variable</td>
</tr>
<tr>
<td>$l_t$</td>
<td>Turbulent length scale</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Turbulence viscosity</td>
</tr>
</tbody>
</table>
\( \tau \) = Shear Stress (Wall)

\( k \) = Turbulence Kinetic Energy

\( \varepsilon \) = Epsilon (Turbulence Eddy dissipation)

\( \varepsilon_{nb} \) = Solid blockage correction factor

\( \varepsilon_{wb} \) = Wake blockage correction factor

\( \omega \) = Turbulence Eddy Frequency

\( \delta \) = The identity matrix or Kronecker delta Function.

\( \Gamma \) = Diffusivity.

\( \kappa \) = Von Karman constant.

\( p' \) = Modified pressure

\( \mu \) = Dynamic viscosity

\( \phi \) = General scalar variable

\( \otimes \) = Tensor product, an operation which is a Multiplication of one vector by another

\( \nabla, \nabla \cdot \) = Vector operators
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>CALTRANS</td>
<td>California Dept. of Transportation</td>
</tr>
<tr>
<td>OSD</td>
<td>Office of Structures Design-(CALTRANS)</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CPU</td>
<td>Central Processing Unit</td>
</tr>
<tr>
<td>MeTiS</td>
<td>Multilevel Partitioning Algorithm Software</td>
</tr>
</tbody>
</table>
Introduction

1.0 Introduction

The design of bluff bodies such as long span bridge deck sections, involves both complex fluid – structure interaction, dominated by aeroelastic stability considerations, flow patterns of the fluid motivated and guided by the shapes of objects attached to the bluff body section, as well as the structural response of the bluff body itself.

Until recently, aerodynamic / aeroelastic characteristics of such sections were obtained almost exclusively by sectional model test, taut-strip model or full scale model testing in a wind tunnel laboratory. Often, these tests are aided or done in conjunction with water channel or smoke experiments to aid with the visualization and highlight the effects of the fluid – structure interaction.

The design process of a long span bridge deck section is often left in the hands of Structural Engineers with a Bridge design specialty; and on rare occasions, architects are sometimes involved. The design process involves using well known rules of thumb such as depth to span ratios to establish the depth of the super structure, while the shapes and types of parapets are often dictated exclusively by the safety of vehicles and pedestrian traffic for which the bridge will be used for, rather than a combination of safety, aesthetics and aerodynamic considerations. Once the basic deck section has been established, the next stage is to ensure that it can handle the anticipated load and the
accompanying stresses from live load caused by defined standardized vehicles, dead or permanent load including the self weight of the structure, horizontal wind load as well as seismic loads in most cases. The choice of the depth of the deck section is often at the selection of the engineer, while the width of the deck is governed by vehicular traffic needs such as lane widths and or pedestrian access. Current practice in the choice of edge details is a combination of established practice i.e. a design based on prior bridge deck sections that have been proven to work with some optimization within design codes. Figure 1a-e shows various edge details of long span bridge decks from different parts of the world, most look similar with sharp pointed edges with the exception of the Normandie Bridge, yet there is variation in the geometry and how the bottom flange plates are shaped and connected to the exterior girder to form the edge details. The Normandie Bridge is curiously at variance in its edge details, while the Little Belt Bridge and the Tsing Ma Bridge makes no effort to integrate the bottom flange plates in an aerodynamic fashion but relying only on the immediate edge as the primary aerodynamic controlling mechanism. All of this suggest and shows variation in design philosophy. It also lends credence to the philosophy of design based on established practice.

Once the basic sectional geometry has been established, then wind tunnel test of the bridge structure become mandatory as a practical matter of fulfilling mandated code requirements so as to obtain the aerodynamic characteristics. After establishing that the deck section is aerodynamically acceptable, it is common that modifications or changes to any elements on the bridge either for aesthetics or structural reasons would necessitate further wind tunnel test to make certain that such modifications have not or will not have
an adverse effect on the aerodynamic characteristics of the bridge. The process is often long and expensive; hence a numerical procedure that is economical, and which could also eliminate some of the initial wind tunnel test(s) during the preliminary design stages is desirable. The aim of CFD then is to lead the design process towards its optimization in an economical way.

The mathematical theory underlying the numerical simulation and studies of the aerodynamic behavior of long span bridge deck sections are primarily based on the partial differential equations (PDE) that describe fluid flow. The techniques that have been used in the last decade to solve partial differential equations (PDE’s) include:

- The Finite Difference Method.
- The Finite Volume Method.
- The Finite Element Method.
- The Discrete Vortex Method – Which is not grid based.

Just as the collapse of the Tacoma Narrows Bridge encouraged research and refined methodologies in long span bridge design, so the proportional reduction in computational cost encouraged the use and application of computational fluid dynamics for numerical solutions of long span bridge deck sections.

The goal of Computational Fluid Dynamics (CFD), and Computational Field simulations in general, is to provide answers to engineering problems using computational methods to simulate fluid physics. CFD usage in many industrial
applications has demonstrated the capability to predict trends for modifications and parametric design studies. Its most valuable contribution today may be in allowing detailed understanding of the flowfield so as to determine causes of a specific phenomenon.

(a) Messina Strait Bridge - Brown (1993)

(b) Tsing Ma Bridge - Zhu (2002)
Figs. 1a-e cont. Variation in edge details for long span bridge deck sections.
1.1 Scope of the work

The study of the effects of edge details in relationship with the aerodynamic characteristics of long span bridges has not received much attention from designers and researchers alike. While the use of CFD programs has been applied to bluff section of the long span deck section type, the sections studied so far have been greatly simplified two-dimensional models that do not fully take into consideration the details pertinent to real bridges. There are a number of classes of CFD that might be applied but attention here is focused on the methods most commonly used in engineering applications for investigation of three-dimensional problems. Three dimensional because nearly all real problems in bridge aerodynamics are three dimensional. The chosen methodology must also satisfy conditions of robustness and manageable computing resource requirements. The primary aim then of this work is the application of CFD using RANS (Reynolds Averaged Navier–Stokes) models to the study of three dimensional bridge aerodynamics. In this regard, the objective is to study and understand what is feasible and good practice in the application of RANS CFD. Further, the dynamics of the bridge will not be included in the analysis, so there is no coupling of this to the fluid flow- this is an obvious topic for the continuation of the research. A specific bridge is chosen for the case studies, the Carquinez Strait Bridge in California; a design is clearly needed and using a recently built bridge is considered to be preferable to adopting a generic design. Therefore this study aims to understand:

- Whether the flow around slender prismatic structures of the long span bridge deck section type composed of two-dimensional cross section is three
dimensional. Sectional wind tunnel test flows are essentially two-dimensional, however, the presence of equipment made of such elements as posts spaced in both direction on the deck might produce three-dimensional effects that has bearing on the aerodynamic properties such as separation and reattachment points, and location and forms of eddies on the deck.

- The relationship between edge details and the aerodynamic characteristics of long span bridge deck sections. The steady forces, such as lift, drag and pitching moment will be compared with wind tunnel tests (sectional) with variations with and without equipment such as parapets, barriers and the variations of the leading and trailing edges of various deck sections while keeping the deck width constant.

- Whether numerical simulations can lead to a reduction on the number of expensive physical model tests that are currently required to determine the optimum aerodynamic properties for a typical cross-section of a single structure.

- The different modes of vortex shedding attributable to different leading/trailing edge types are investigated in this research. The stability of long span bridges is to a large extent dependent on the vortex induced oscillations as seen in the case of the Tacoma Narrows Bridge. Firstly, the original Tacoma Narrows did undergo vortex induced vibration (VIV) in its
short life, but its failure mode was self excited torsional flutter instability. Secondly, bridge designers are concerned about VIV, but they have much more severe concern about flutter, particularly on long span bridges. It is anticipated that the trailing and leading edge shapes will play significant roles in this phenomenon, and therefore an understanding of the mechanism is desirable.

As of now, the aim of CFD is to complement the wind tunnel test and not to replace it, primarily because none of the turbulence models currently implemented in most CFD software can consistently answer all the pertinent questions that arise in the design of long span bridge decks. Therefore, the role of CFD is to work through the preliminary design stages and answer the relevant 'what if' questions that often arise during this stage. As stated earlier, most of the previous work has been done with two-dimensional models, however, the optimization of edge geometry methods must be able to treat three-dimensionality because of the functional purpose of traffic and pedestrian barriers as well as equipment that are often located on the deck. It is not uncommon that the barriers on the deck whether for traffic or pedestrian are often different from one edge to the other, also, permanent equipment that for example is used for maintenance may be located on just one edge of the deck.

Turbulence models such as the Large Eddy Simulation (LES) and the grid less Direct Vortex Method (DVM) although discussed are not really practical for this work. The LES is prohibitively expensive in both time and computational resources, while the DVM is
very attractive, its usage is not so widespread yet and its not commercially available and its limited to 2-dimensional problems.

The issue of vortex induced vibration is of great importance in the design of long span bridge decks, obviously this requires a dynamic model. Unfortunately, a CFD software must be coupled to a Finite Element Method software for structural analysis using specialized interface program to permit this type of study, while it could be done, it is very expensive and requires very large resources and this was not within the scope of this work and would obviously be a reasonable and obvious follow up to this work as the costs of computing decreases.

1.2 Organization of the thesis

Following this introductory chapter is the literature review, essentially a review of relevant work that is related to bridge deck sections, parapets or similar objects that may be found interacting with or as part of the main bridge deck section, turbulence modeling and effects of edge details on flow behavior. Chapter 3 reviews the basic turbulence model and advection schemes that are available in most commercial CFD solvers. Here a deliberate attempt is made to not invoke the characteristics and or usability or promote any particular software. This author's investigation shows that virtually all the software available is reasonably identical in their formulation of the turbulence models. In essence, there is neither a monopoly of knowledge nor proprietary scheme unique to any program and therefore it's meaningless to dwell on any particular software. The chapter concludes with a parametric study of the most widely used and generally available turbulence
models, with a recommendation on the most suitable for studying external flow of the kind and type encountered in this research.

Chapter 4 dwells on grid generation processes and the methodology used in this research, it concludes by examining via a parametric study, the most accurate mesh element type for modeling the deck sections.

Chapter 5 explains the mechanisms and the computational efforts necessary to be able to simulate the created model. It is also an economic issue since the computational costs can be estimated in comparison to wind tunnel tests.

Chapter 6 is concerned with numerical simulation; encompassing the boundary condition and the physics of the simulation (in CFD terminology the process of describing, the boundary condition and characteristics of flow.) The geometry of the deck sections investigated and the related wind tunnel tests are not public information, they are and remain confidential properties of the state and government of the state of California. This author was given access and use of the wind tunnel results under the strictest condition for its use and publication in this research.

In chapter 7, a review of the results and the effects of edge details on the four prototype deck sections is given. The thesis concludes in Chapter 8 with a summary and recommendation for future work. Table 1 summarizes the major activities involved in this work and the relevant location in the thesis.
<table>
<thead>
<tr>
<th>Stage</th>
<th>Activity</th>
<th>Chapter</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Problem Definition.</td>
<td>Three Dimensional flow over a fixed, rigid bridge deck. Case studies on the Carquinez Straight Bridge</td>
<td>Chapter 6</td>
</tr>
<tr>
<td>2. Solution Strategy.</td>
<td>Review and test (where appropriate) potential strategies. Select one that is well founded in engineering use, and is suitable for the chosen application and consistent with typical computing resources and establish solution strategy.</td>
<td>Chapter 3</td>
</tr>
<tr>
<td>4. Turbulence Models.</td>
<td>Review available models (from methods such as DNS to one-equation turbulence model) Select and test the appropriate sub-set and establish the preferred model.</td>
<td>Chapter 3</td>
</tr>
<tr>
<td>5. Geometry.</td>
<td>Define test geometries based around the Carquinez Straight Bridge design.</td>
<td>Chapter 6</td>
</tr>
<tr>
<td>6. Grid Design.</td>
<td>Review grid design procedures. Test sensitivity to grid design and select appropriate grid design.</td>
<td>Chapter 4</td>
</tr>
<tr>
<td>8. Initial Conditions</td>
<td>Define inlet and initial conditions based on the wind tunnel conditions and select appropriate inlet</td>
<td>Chapter 6</td>
</tr>
<tr>
<td>Chapter 3 &amp; 4</td>
<td>10. Applications</td>
<td>Run series of case study calculations using the established methodology.</td>
</tr>
<tr>
<td>Chapter 7</td>
<td>11. Post Processing</td>
<td>Analyze output, review performance.</td>
</tr>
<tr>
<td>Chapter 8</td>
<td>Propose application guidelines</td>
<td></td>
</tr>
<tr>
<td>Chapter 8</td>
<td>Recommendations for future research.</td>
<td></td>
</tr>
</tbody>
</table>

*Table 1. Summary of Research Activity.*
1.3 Summary

This chapter reviews the methodologies and the application of CFD in the analysis of long span bridges. From the four different deck sections shown in figure 1, it is evident that there is no uniformity in approach and design of the edge details. While the width of the deck section is controlled by traffic/pedestrian usage, the edge details does not have a code mandated criteria for design, nor is there a clear understanding of why one edge type is preferred over another, hence the need for this research.

The scope of work has also been laid out, the tool for this work is essentially commercial CFD package. While the application of CFD into Civil Engineering structures is still in its infancy, the usage for similar type structure such as the airfoil has gained widespread acceptance within the engineering community at large. It is now commonly accepted that CFD can be used to validate and or design various types of structures. It is expected that CFD will play an increasing role complementary to the wind tunnel testing, and become widely used within the bridge design community in studying what if scenarios.
2.0 Introduction.

The witnessed collapse of the Tacoma Narrows Bridge in 1940 (figures 2a & 2b) at a wind speed of 19m/s inspired and encouraged theoretical, experimental and numerical simulation in Aerodynamics of Long Span Bridge Deck sections. In an exhaustive report on the failure of the Tacoma Narrows Bridge, Farquharson (1949) detailed technical aspects of long span bridge deck design including experimental techniques, aerodynamics and structural dynamics. The report advanced the concept of sectional bridge deck models supported on elastic springs that could be used to obtain vortex shedding excitation and flutter instability characteristics of suspension bridges.

Scanlan and Sabzevari (1969), using the parallelism between an airfoil and a bridge deck section and using similar experimental setup as advanced by Farquharson derived the basic mathematical relationship of the so-called flutter derivatives relating the interaction between the aerodynamic force and moment of an oscillating bridge deck section. The development of this link and subsequent definition and redefinition by several researchers of the static and dynamic interaction between a bluff body and unsteady fluid dynamics forms the basis of Computational Wind Engineering as applied to Long Span Bridge deck design.

Cross-sectional shape including attachments is an important parameter in Aeroelastic
stability considerations in the design of Long span bridge deck sections. According to various authors, the effect of section details cannot be disregarded in Bridge Aerodynamics. Using wind tunnel tests, Bienkiewicz (1987), Nagao et al. (1993) showed the influence of partial streamlining and traffic barriers on the vortex induced response of bridge decks. Using the same strategy, Scanlan et al. (1995) showed the critical dependency of bridge flutter derivatives on details such as deck parapets. In addition, Scanlan considers the modeling of the section details as a critical part in sectional wind tunnel tests as a function of accuracy. While for a long time wind tunnel tests provided the basis for the design, numerical simulations have been developed to the extent that the essential practical design criteria can be assessed, simulated and studied. Careful research is therefore needed in applying these numerical methods.

In the following literature review, only contributions relevant to bluff body relating to bridge aerodynamics, in the context of Computational Fluid Dynamics including solution strategies and their limitations, flow patterns including the physics definition, meshing and grid types and edge detail considerations are reviewed.
Fig. 2a Torsional Mode of the Tacoma Narrows Bridge - Farquharson (1949)

Fig. 2b Failure of the Tacoma Narrows Bridge - Farquharson (1949)
2.1 Computational Wind Engineering.

According to Murakami (1998), Computational Wind Engineering (CWE) is defined as Computational Fluid Dynamics (CFD) applied to Wind Engineering. A study in bluff body aerodynamics essentially begins with the airfoil as the basis of definition of aerodynamics characteristics and the relevant terms and equations. In 2-D flutter studies, Brar et al. (1996) suggests that the fluid-structure interaction problem can be simplified by considering the 2-D airflow about a representative section along the span of a long and flexible structure. Consider figure 2.1a from Scanlan (1975) which shows such a generic section, where $b$ is the semi chord, $ab$ is the distance between the point of rotation (torsional axis) and midchord, $h$ is the heaving or bending deflection of the rotation point and $\alpha$ the angular deflection about the rotation point. For a symmetrical airfoil supported about its geometric center or a bridge deck section having its torsional axis at the midchord ($\alpha=0$) as shown in figure 2.1a, the equations of motion are derived by Scanlan & Rosenbaum (1968) as:

\[
\begin{align*}
    m \left[ \ddot{h} + 2\zeta_h \omega_h \dot{h} + \omega_h^2 h \right] &= L_h \\
    I \left[ \ddot{\alpha} + 2\zeta_{\alpha} \omega_{\alpha} \dot{\alpha} + \omega_{\alpha}^2 \alpha \right] &= M_\alpha
\end{align*}
\]

where $m =$ mass per unit span of structure, $\zeta_h =$ damping, $h =$ heaving displacement, $\dot{h}, \ddot{h} =$ heaving velocity & acceleration respectively, $\omega_h, \omega_{\alpha} =$ frequency in $h$ and $\alpha$ degrees of freedom respectively and $L_h, M_\alpha =$ aerodynamic lift and moment about elastic axis respectively.

The study of bluff body aerodynamics while distinct, shares many similar concepts from airfoil flutter theory and hence many researchers start from the airfoil concept. Consider a
thin symmetrical airfoil in a uniform airflow at an incident angle of $\alpha$, the lift based on potential flow theory is given as (Brar et al 1996):

$$L = \frac{1}{2} \rho U^2 B \frac{dC_L}{d\alpha}$$  \hspace{1cm} (1.3)

where, $C_L$ is the lift coefficient per unit span, and $\frac{dC_L}{d\alpha} = 2\pi$ where the boundary layer and wake are considered negligibly small, and $U$ is the oncoming uniform flow velocity without any turbulence and $B$ is the chord length. These expressions are derived according to Pantom (1984). In the ideal flow theory, lift is dependent on the circulation of the fluid around the body which in turn is determined by satisfying the Kutta condition at the trailing edge. Kuethe (1959) defines the Kutta condition as the circulation created by a body with a sharp trailing edge around itself is of enough strength to hold the rear stagnation point at the trailing edge. If the motion is unsteady, then the airfoil influences the motion around it and the lift is no longer a simple function of the circulation. This non-uniform motion creates aerodynamic inertia forces and the circulation is accompanied by vortex shedding from the trailing edge. Hence, the lift will be a function of the motion and the geometry of the section.
The aerodynamic forces on an oscillating airfoil in a uniform airflow of velocity $U$, was determined by Theodorsen (1935), the lift and the moment forces are given as:

\[
L = -\frac{1}{2} \rho U^2 b (2\pi) \left( \frac{b \alpha}{U} + \frac{b^2}{U^2} \right) \left( \frac{1}{2} - a \right) U \left\{ \frac{b \alpha}{U} \right\}
\]

\[
M = -\frac{1}{2} \rho U^2 b^2 (2\pi) \left( \frac{1}{2} - a \right) \frac{b \alpha}{U} - \frac{b^2}{U^2} a h + \frac{b^2}{U^2} \left( \frac{1}{8} + a^2 \right) \alpha + 2C \left( a + \frac{1}{2} \right) \left[ \frac{\alpha + h}{U} + \left( \frac{1}{2} - a \right) \frac{b \alpha}{U} \right]
\]

where $C = C(k)$, is the so called complex Theodorsen circulation function, and is given as

\[
C(k) = F(k) + iG(k)
\]

where $k = \frac{b \omega}{U}$, $b = \frac{B}{2}$ is the half-chord of the airfoil, and $\omega$ is the frequency of the oscillatory motion.
Figure 2.1b shows the plots of the components of the functions $F$ and $G$ of the
Theodorsen circulation function which are defined by Bessel function. It is evident that
the circulatory part of the Theodorsen function is dependent on the effective angle of
attack, measured from the rearward three quarter chord point of the airfoil which is
unique to thin airfoil theory.

\[ \alpha_{\gamma_4} = \left[ \alpha + \frac{h}{U} + \left( \frac{1}{2} - a \right) \frac{b \alpha}{U} \right] \]  

(1.6)
Wagner (1925) determined the indicial response of an airfoil to a step change in angle of attack, which is another way of determining the above mentioned circulatory lift. Wagner's proposal is based on the potential flow theory (Garrick, 1938), it states that the lift function with time for a theoretical flat plate airfoil due to an impulsive change in angle of attack, \( \alpha_0 \) as:

\[
L = \frac{1}{2} \rho U^2 2b (2\pi) \alpha_0 \phi(s)
\]

where \( s = Ut/b \) is the dimensionless time and \( \phi(s) \) is the so-called Wagner function approximately given by:

\[
\phi(s) = 1 - 0.165e^{0.0415s} - 0.335e^{-0.300s}
\]

(1.7)

For a thin airfoil, the effects of the \( \alpha \), \( \dot{\alpha} \) and \( \dot{h} \) motions are inclusive in \( \alpha_{y4}(s) \) function given in equation 1.6, it follows than the circulatory lift is given by superposition as:

\[
L(s) = -\frac{1}{2} \rho U^2 2b (2\pi) \int_0^s \alpha_{y4}(\sigma) \phi(s - \sigma) d\sigma
\]

(1.9)

where:

\[
\alpha_{y4} = \frac{d}{ds} \alpha_{y4}(s)
\]

(1.10)

Since long span bridge sections are relatively bluff compared with an airfoil, bridge flutter is not necessarily resolved with potential flow theory. Scanlan (1971) suggested the following expressions for bluff body oscillatory motion which are in common use today:
\[ L = \frac{1}{2} \rho U^2 2b \left( kH_1^* \frac{\dot{h}}{U} + kH_2^* \frac{b\dot{\alpha}}{U} + k^2 H_3^* \alpha + k^2 H_4^* \frac{\dot{h}}{b} \right) \]

\[ M = \frac{1}{2} \rho U^2 b^2 \left( kA_1^* \frac{\dot{h}}{U} + kA_2^* \frac{b\dot{\alpha}}{U} + k^2 A_3^* \alpha + k^2 A_4^* \frac{\dot{h}}{b} \right) \]

and \[ H_1^* = 2\pi \frac{F}{k}, \quad H_2^* = -2\pi \left( \frac{1}{2} + \frac{G}{k} + \frac{F}{2} \right), \quad H_3^* = -2\pi \frac{F - k \frac{G}{2}}{K^2}, \quad H_4^* = 2\pi \left( \frac{1}{2} + \frac{G}{k} \right) \]

\[ A_1^* = \frac{\pi F}{k}, \quad A_2^* = \frac{\pi}{k} \left( \frac{G}{k} + \frac{F}{2} - \frac{1}{2} \right), \quad A_3^* = \frac{\pi}{k^2} \left( F - k \frac{G}{2} + \frac{k^2}{8} \right), \quad A_4^* = -\frac{\pi G}{k} \]

where \( h \) is the heaving displacement, \( \alpha \) the torsional displacement, \( k \) the reduced frequency, \( \rho \) the air density and \( b \) the half chord length.

In comparison with equation (1.3), the coefficients \( kH_2^* \) and \( kA_2^* \) are analogous to force coefficient gradients.

Ming Gu et al. (2000) proposed an identification method to extract all the flutter derivatives as proposed by Scanlan (1971). Their method is based on the signals of the coupled vertical-torsional free vibration of the spring suspended section model. The procedure relies on a unifying least squares theory. In this method, a unified error function, made up of linearly combined error functions of vertical and torsional motions, are defined to optimize the flutter derivatives. Equations (1.1) & (1.2) are rewritten in matrix style as:

\[ [M] \{\ddot{x}\} + [C] \{\dot{x}\} + [K] \{x\} = \{F_m\} \] (1.12)
where \([M], [C], \) and \([K]\) are the mass, damping and stiffness matrices, respectively, of the bridge, and \(\{F_{se}\}\) is the self excited forcing vector. According to the complex mode parameters, the damping and stiffness matrices in equation (1.12) are thus written as:

\[
\begin{align*}
[C^e] &= [M]^{-1} [C] \\
[C] &= [M]^{-1} [C] \\
[K] &= [M]^{-1} [K]
\end{align*}
\]  

(1.13)  

(1.14)  

(1.15)  

(1.16)

where the superscript is used to denote the (wind-bridge) system’s stiffness and damping matrices. The flutter derivatives are finally obtained from the above matrices as follows:

\[
\begin{align*}
H_1^* &= -\frac{m}{\rho B^2 \omega} (\bar{C}_{11} - \bar{C}_{11}), \\
H_2^* &= -\frac{m}{\rho B^2 \omega} (\bar{C}_{12} - \bar{C}_{21}), \\
H_3^* &= \frac{m}{\rho B^2 \omega^2} (\bar{K}_{12} - \bar{K}_{12}) \\
H_4^* &= \frac{m}{\rho B^2 \omega^2} (\bar{K}_{11} - \bar{K}_{11}) \\
A_1^* &= \frac{I}{\rho B^2 \omega} (\bar{C}_{21} - \bar{C}_{21}), \\
A_2^* &= \frac{I}{\rho B^2 \omega} (\bar{C}_{22} - \bar{C}_{22}), \\
A_3^* &= \frac{I}{\rho B^2 \omega^2} (\bar{K}_{22} - \bar{K}_{22}) \\
A_4^* &= \frac{I}{\rho B^2 \omega^2} (\bar{K}_{21} + \bar{K}_{21})
\end{align*}
\]

2.2 Flutter

Collectively, \(H_1^*, \ldots, H_4^*, A_1^*, \ldots, A_4^*\) from equation (1.11) are non-dimensional and are termed aerodynamic (flutter) derivatives. These aerodynamic derivatives must be obtained by wind tunnel experiments or by numerical flow simulations. For long span
bridge deck sections, one of the main Aeroelastic effects of concern is flutter, see Astiz (1996) and Larsen & Walter (1996). The susceptibility of a deck section to flutter instability depends on the magnitude and sign of the flutter derivatives, if for example $H_1^*$ is positive then galloping will occur, if $A_2^*$ is positive then stall flutter will occur, and if both $H_2^*$ and $A_1^*$ are positive then classical flutter will dominate.

According to Drybye & Hansen (1996) flutter occurs at a critical wind speed at which the energy input from the motion-induced wind load is equal to the energy dissipated by structural damping. The critical wind velocity is called the flutter wind velocity and it typically occurs at a high wind velocity ($\approx 70 \text{ m/s}$) see Frandsen (2004).

The flutter phenomenon was first investigated in aerospace engineering and the relevant terms were then carried over to wind engineering. Flutter of bridge deck sections are described by Simiu & Scanlan (1986) and Larsen and Walther (1996), and can be characterized as follows:

- **Single degree of flutter in torsion**, also-called stall flutter, is a pure torsional motion of the bridge section. In this instance, the amplitude of the torsional oscillations grows with increasing velocity.

- **Binary flutter**, also known as classical flutter, is a coupled vertical and torsional motion of the bridge section. Once the wind velocity exceeds the flutter wind velocity, the oscillations grows often to catastrophic amplitude and proportions.
Classical flutter often occurs at a wind speed where the motion induced wind forces creating vertical and torsional vibration modes that are coupled. Flutter instability occurs at higher wind speeds because of the dominant self excited accompanying aerodynamic forces. These forces are characterized by torsional motions, and may be accompanied by vertical bending motions. Figure 2.3b shows the characteristic response due to classical flutter that involves a 2-Dimensional bluff section moving albeit with restraints in both vertical translation and rotation. Frandsen states that "the potential energy input by the aerodynamic forces in flutter is very large and consequently the rise in amplitude is known to be rapid if the flutter limit is reached with catastrophic effects". In contrast to vortex induced vibrations, this phenomenon is also insensitive to structural damping as shown in figure 2.3b. As described by Simiu & Scanlan (1986), flutter may involve non-linear aerodynamics behavior. However, the onset flutter problem has been successfully treated by linear analysis methods.

Wind tunnel tests typically provide a lot of information of great importance in long span bridge design in regard to flutter and the sensitivity associated with changes in the leading edge geometry. Parkinson (1989) suggests that the influence of this sensitivity(s) cannot be predicted by semi-empirical analytical models such as lift or wake-oscillation modeling. In addition, quite a number of sectional model tests are often required, Larsen & Jacobsen (1992) reported of 16 such tests carried out on the Great Belt East Bridge (GBEB), with each test taking up to 8 weeks to complete, a rather daunting task just for the preliminary design stages.
Frandsen (2004) investigated the use of fluid and structural finite elements to predict classical flutter, using the Great Belt East Bridge as a case study (fig 2.3a). The fluid-structure interaction study was idealized with two-dimensional models using the Arbitrary Lagrangian-Eulerian (ALE) finite element procedure. In the ALE methodology, fluid particles are allowed to move independently of the structural motion, in addition an unsteady laminar flow model was assumed and the bridge deck section was without railings or parapets. In these studies, the fluid flows were first simulated around the bridge deck section that is held in a stationary position. Further studies involving the sensitivity of the mesh requirements were carried out, while the predicted static forces of lift and drag as well as the Strouhal number were then compared with wind tunnel test results. Next, the bridge deck was then coupled with the fluid model, while the first natural and torsional frequency as well as mass moment of inertia of the full-scale structure prescribed so as to predict the flutter derivatives.

**Fig 2.3a** Deck model of the Great Belt East Bridge used in FE Simulations for flutter – *Frandsen (2004)*
Frandsen (2003) also suggested that the development of flutter is not dependent on the higher form of frequency of bluff body vortex shedding characteristic. This may be the case for long span bridges, where the two Aeroelastic phenomena occur at very different wind speeds, but may not be the case for small, more flexible bridges such as footbridges. In these cases, VIV (vortex induced vibration) and flutter may not be completely independent, occurring at similar wind speeds, and may even be coupled in some way with VIV response causing flutter instability to be initiated as it was with the Tacoma Narrows Bridge. Furthermore, the flat plate flutter theory of Theodorsen (1935) was confirmed although inviscid flow was assumed, indicating that accurate modeling of the boundary layer may not be as critical for this type of Aeroelastic phenomena as in the modeling of vortex induced vibrations. The modeling and prediction of flutter then seems to be mainly affected by the leading edge separations and the associated pressure forces. As for the flat plate theory itself, it is limited to streamlined deck sections, where it is a reasonable approximation.

For bluffer sections, it does not give accurate results, and cannot capture the torsional flutter instability as that experienced on the Tacoma Narrows. Other researchers such as Jenssen and Kvamsdal (1999) performed similar studies but using the Finite Volume Method (FVM) to model the flow field on moving unstructured grids. Still the flutter limits were based on 2-D analysis with prescribed deck motions, as opposed to self-excited motions; however the flutter limits were in good agreement with those obtained from wind tunnel tests. Using a combination of the finite-difference and the vortex-method scheme, Brar (1997) developed a model where an Eulerian finite
difference grid was located in the viscous sub-region next to the bluff body of interest as well as the Langragian vortex element domain in the free stream region away from the wall boundaries. The Reynolds number ranges from 100-1000 with the Strouhal number predictions from the work were in good agreement with other similar research studies of the same problem.

Theodorsen (1935) developed the solution for the onset of flutter using the 2-D inviscid flat-plate theory. Later, Selberg (1961) used the same theory to determine the same for a bridge deck section as:

\[
\frac{V_f}{f_a B} = 5.246 \sqrt{\left( \frac{m}{\rho_{el} B^2} \right) \left( 1 - \left( \frac{f_x}{f_a} \right)^2 \right)}
\]  

(1.17)
where $V_f$ is the critical flutter wind velocity, $m$ is the mass, $r = \sqrt{I_m/m}$ is the radius of gyration, $I_m$ is the mass moment of inertia and $B$ is the width of the bridge. $f_v$ and $f_a$ are the vertical natural frequency and the torsional natural frequency respectively.

Frandsen (2004) also suggests that both the vertical natural frequency and the torsional natural frequency of the structure are important in determining the critical speed for the onset of flutter. When the ratio of $f_v/f_a$ is less than 1, Selberg’s equation is known to be reliable in its predictions of the speed of the onset of flutter for trapezoidal box girder shapes.

2.3 Other Types of Aeroelastic Phenomena

Apart from flutter, there are other aeroelastic phenomena that are observed in bluff body aerodynamics. There are three that are of great importance in long span bridge sections because they involve separation of flow from the body and thus cause periodic excitation and invariably instabilities which can result in catastrophic failure. Separation is not necessary for the occurrence of flutter, which is observed in airfoils specifically designed to avoid flow separation.

The three major ones are:

- Galloping
- Buffeting and
- Vortex induced vibrations.
Galloping is a form of Aeroelastic instability produced when the aerodynamic damping is not positive over part or all of the oscillatory cycle of the deck section. It is characterized by pure translational crosswind motion of the structure; however, it can also occur in the along-wind degree of freedom, depending on the cross-section of the object. Galloping is often accompanied with large amplitudes of oscillations at frequencies that are lower than the natural frequency of the structure and it is typical of slender structure.

Buffeting is an induced forced vibration of a structure due to fluctuations in the oncoming wind speed, it may also result from the interference of a second body situated in the wake of the first (Havel et al, 2001). For the bridge deck section, this can lead to its vibration albeit low, its real effect is psychological on drivers of vehicles that might be on the bridge at the time, because objects attached to the bridge such as parapets and barriers can create wakes causing turbulent eddies thereby making driving a hazard.

2.4 Vortex Induced Vibrations (VIV)

Vortex induced vibrations can affect bluff bodies, such as bridges. Turbulent and vortices usually accompanies the flow around bluff bodies; vortices usually in the wake as a result of the wind creating pressures on the windward side and suction on the leeward side. The excitations produced in bluff body aerodynamics are various, and it is worthwhile to
classify them. One widely accepted classification is that proposed by Naudascher and Rockwell (1994), consisting of 3 different types of such excitation:

- **EIE:** Extraneously induced excitation created from the periodic pulsation of oncoming flow or by turbulent buffeting.

- **IIE:** Instability induced excitation; a form of excitation induced by the von Karman street created as a result of flow instability inherent to the flow created by the structure under consideration.

- **MIE:** Movement-induced excitation; arising from the movement of the body as a result of the forces exerted by the fluid.

It is possible that all of these phenomena can act simultaneously. Naudascher & Wang (1993) further classified the IIE based on the type of wake formed behind the bodies:

- **LEVS:** Leading-edge vortex shedding (flow separation at the leading edge and formation of vortices which dominate the near wake of the body).

- **ILEV:** Impinging leading-edge vortices (flow separation at the leading edge and impingement of the leading edge vortices at the side surfaces and/or edges of the body).

- **TEVS:** Trailing-edge vortex shedding (decisive flow separation at the trailing edge and vortex shedding analogous to the von Karman Street behind circular cylinders).
• AEVS: Alternate-edge vortex shedding (vortex shedding occurring at a critical range of incidence, where an alternating vortex shedding occurs; one vortex separates at the leading-edge and the other at the trailing edge).

The development of vortex shedding can be explained if viscosity is considered. Only a viscous fluid will satisfy the no slip condition on the solid surface of a body immersed in its flow. Even at a low level of this condition, it will still hold, only that its influence is confined to a rather small region: the boundary layer region along the body. At high Reynolds number the boundary layer is thin; i.e. the influence of viscosity is confined. Boundary layers were first postulated by Prandtl (1934); the boundary layer can be seen for an airfoil and a flat plate as shown in figure 2.4a.

![Fig. 2.4a Examples of Boundary Layers on (a) Airfoil and (b) Plate](Pictures from Iowa Institute of Hydraulic Research, University of Iowa, Ames Iowa USA)

As shown in (fig. 2.4b), within the boundary layer, the velocity of the fluid changes from zero on the surface to the free stream of the flow section.
As stated before, the fluid viscosity causes a transfer of momentum from one fluid particle to another, causing a slowdown on those particles closest to the surface, correspondingly to the friction drag force, as shown in figure 2.4c.
At surfaces with high curvature there can also be an adverse pressure gradient adding to the retarding action, which may cause the flow to be interrupted entirely and cause a detachment of the boundary layer from the wall. This is termed separation.

Given the characteristics of the process involved in the separation process, it is understood that viscosity and free stream velocity have an important influence and can be described by the Reynolds number, expressed as:

$$\text{Re} = \frac{UL}{\nu}$$  \hspace{1cm} (1.18)

where, $$\nu = \frac{\mu}{\rho}$$ is the kinematic viscosity and $$l$$ the characteristic length. It should be noted that outside the boundary layer, the flow is independent of the Reynolds number. The Reynolds number then expresses the ratio between the inertia force and the friction force acting on the fluid.

Studies on flow past cylinders, shows a great variety of changes in the flow patterns depending on the variation of the Reynolds number, the dependence of the drag coefficient on the Reynolds number is shown in figure 2.4h.
Fig 24h Reynolds number’s influence on Drag Coefficients for circular cylinder from Acheson (1995) Where $C_d$ is the drag coefficient and $2a$ is the characteristic length and $v$ is the viscosity.

It is well known that vortex shedding exerts a fluctuating force on a bluff body. Strouhal (1878) defined a dimensionless shedding frequency, the so-called Strouhal number defined as:

$$St = \frac{f d}{U}$$  \hspace{1cm} \text{(1.19)}$$

where $f$ is the shedding frequency and $d$ the across-flow dimensions of the body.

Since Strouhal’s investigation, other researchers have found the shedding to be highly dependent on the cross-sectional geometry of the body and consequently have attempted to define a generic Strouhal number, which will be independent of the geometry. Roshko (1954) proposed such a type of universal Strouhal number. Roshko performed experiments on flow past a circular cylinder, and from the wake width, proposed a notched hodograph theory which is then used to define the Strouhal number as:
where \( K = \sqrt{1 - C_{pb}} \) is the velocity along the free streamline relative to that of the uniform on coming flow and \( D' \) is the lateral distance between the two free stream lines obtained from Roshko’s notched hodograph theory. This ‘universal’ Strouhal formula was deficient and limited to a circular cylinder, a normal plate and some bluff bodies. It should also be noted that Nakamura’s (1996) experimental studies described below, rendered Roshko’s universal Strouhal number inapplicable, primarily due to the understanding of the effect of an after body, the presence of which alters the structure of the vortex formation.

Goldburg et al. (1965) and Nakaguchi et al. (1968) also proposed a universal Strouhal type number that both used certain geometrical characteristics of the wake and its formation as a characteristic length scale.

Zdravkovich (1996) presented a comprehensive overview of the vortex shedding process and their different modes on a variety of bluff body sections, with the primary aim of determining the effect of after body shapes on the shedding frequency. It was found that the Strouhal number of a bluff body with after body initially decreases with increasing side ratio, the reduction being independent of the details of after body shape but only on the side ratio. This constitutes a sharp contrast to the base suction that is sensitive to the after body shapes.
Fig. 2.4i, Some Bluff body (2-dimensional) models used in Nakamura's (1996) Wind Tunnel Experiments

Of particular interest in Nakamura's (1996) studies are those sections and shapes shown in figure 2.4i on the previous page, that are most likely to be found in long span bridge design sections or at least close to it. In this, Nakamura reported that the St(D) are nearly the same as shown in figures 2.4j & 2.4k, decreasing with increasing L/D ratio of up
to 1, although the case of a rectangular cylinder departs from the other two when $L/D$ is $> 1.0$ becoming increasingly smaller. If the ratio of $L/D$ becomes greater than 2.0, then the Strouhal number will also increase to a higher value, which corresponds to a kind of impinging shear layer instability; where the two shear layers becomes unstable as a result of the presence of a sharp trailing edge corner.

Strouhal numbers for rectangular, I and H-section cylinders: ○, rectangular; ●, I-section
×, H-section. of figure 2.4j

Fig. 2.4 j-Nakamura (1996)
Strouhal numbers and base suction coefficients for a semicircular and a circular cylinder with splitter plates: $O$, $\bullet$, $St(D)$, semicircular cylinder; $O$, $-C_{pa}$ for semicircular cylinder; $\Delta$, $\bullet$, $St(D)$, circular cylinder; $\Delta$, $-C_{pa}$ for circular cylinder.

Fig. 2.4 k-Nakamura (1996) - Plotted Strouhal number is from Fig 2.4i.

Nakamura further related that the influence of the after body effect on the frequency of the vortex shedding is to reduce it when the spanwise length of the after body increases. If $St^*(D)$ denotes the Strouhal number of a bluff body without after body, then the equivalent Strouhal number $St(D)$ of a bluff body with after body can be expressed as:

$$St(D) = St^*(D) - f\left(\frac{L}{D}\right)$$

(1.21)

here the term $f(L/D)$ is the increasing function of $L/D$. Nakamura assumed then that the function is a kind universal shape modifier that is independent of the geometry of the
cross-section but only on the ratio of $L/D$ when $0 \leq L/D \leq 1.0$

The process of vortex shedding has received interest from a lot of researchers.

Zdravkovich (1996) provided three examples of the form of vortex shedding, stating that the mechanics of vortex shedding are not unique and can undertake a structural change.

There is the so called low-speed vortex shedding, that is related to laminar wake instability and then the high-speed vortex shedding with a distinct mechanism of vortex formation and shedding.

The vortex shedding of the cylinder was studied by Kovasznay (1949), stating that laminar vortices are formed as they are carried gradually downstream and not shed from the cylinder as is commonly thought. This is an example of the low speed mode of vortex shedding. This low speed mode of vortex shedding is attributed to the instability of the laminar wake. As an example, figure 2.4l-1 show the laminar periodic wake at Reynolds number of 140. Here, the distinct features are the sinusoidal trail and gradual roll up of free shear layers at the crests and troughs, and the connection of all the eddies by the trail streak line that is originating near the wake.

Gerrard (1966) defined the high speed mode of vortex shedding as turbulent vortices developed from a stationary position over one half of a shedding period that are strong enough to draw the other shear layer across their wake such that the subsequent vortex is cut-off from a further supply of the circulation.

The characteristic features of the high speed mode of vortex shedding are shown in figure 2.4l-2 at Reynolds number of 300 from Freymuth et el. (1986). Here the trail streamline is not seen, an indication that the vortices are independent and not connected.
The upper vortex that is formed is in an almost static position and cut-off from the free shear layer by the lower vortex. The shedding frequency of the cut-off vortices is determined from the distance between the two free shear layers and the free stream velocity, which is proportional to the free stream velocity.

Fig. 2.4l-1 Laminar periodic wake at Re=140; by Taneda, in Van Dyke (1982)

Fig. 2.4l-2 Periodic wake at Re=300 – Freymuth et. el (1986)
The other type of observed mode of vortex shedding is the synchronized mode, this is observed when a transversely oscillating cylinder for example, takes over the control of frequency of vortex shedding. Griffin and Ramberg (1974) studied the wakes of a forced oscillating cylinder. In Zdravkovich (1982) analysis of the flow (fig. 2.4l-3) visualization photograph from the experiments, two modes of vortex shedding were identified:

- A type of vortex shedding formed on one side of the cylinder while the other side was at its maximum amplitude.

- A second type of vortex shedding formed on one side of the cylinder when it was close to its maximum displacement.

These modes of vortex were found in the lower and upper regions of the synchronization range respectively. Separating the two modes of vortex shedding, is the critical reduced velocity, where there is a discontinuous change in fluctuating and time averaged forces and in the phase angle.

Fig. 2.4l-3 Laminar Synchronized wake behind a mechanically oscillating cylinder, from Griffin & Ramberg (1974)
The third type of observed vortex shedding also from an oscillating cylinder, is a derivative of higher synchronization order when the shedding frequency generated by a streamwise oscillating cylinder is about twice as much. Griffin & Ramberg (1974) further described two additional modes of vortex shedding: one form is characterized by one vortex formed per half-cycle and the other with two vortices forming per each half cycle as shown in figure 2.4m. The transition from the first to the second mode leads to the formation of one vortex in odd half-cycles and a pair during even-half cycles.

Since bluff bodies such as a long span bridge deck sections are subject to fluctuating pressure fields, a complex 3-D fluid structure interaction is developed. Staubli & Deniz (1997) conducted experiments with fluctuating rectangular and octagonal profiles studying the interaction of leading and trailing edge vortex formation. They concluded
that the forces acting on prismatic bodies are directly related to the flow structure and the formation of vortices at the leading and trailing edge. The vortices strongly interact with each other and with the body forces. These effects are very sensitive to geometrical variations such as the elongation ratio of the rectangular profile. Furthermore, frequency analysis of measured lift forces and flow visualization gives evidence that three different classes of vortex formation can be observed with variation of the angle of attack. These are: impinging leading-edge vortices, trailing-edge vortex shedding, and alternate-edge vortex shedding. Using the same classifications as proposed by Naudascher & Rockwell (1994), Naudascher & Wang (1996), compiled a comparison of the effect of body geometry on the process (Fig. 2.4n). From Fig. 2.4n it is observed that there is a sudden jump of Strouhal number occurring at elongation ratios of approximately $L/D \leq 2-3$ and $L/D = 4-7$ marking the three flow regimes as illustrated due to reattachment of the separated flow.

![Fig. 2.4n Classes of Vortex formation observed with increasing elongation of different prismatic bodies. Class I leading-edge vortex shedding; Class II Impinging leading-edge vortices; Class III trailing-edge vortex shedding - Naudascher & Wang (1996).](image-url)
Straatman and Martinuzzi (2003) studied the effect of boundary layer thickness on the onset of vortex shedding from a square cylinder near a wall (figure 2.4d). Their computations were carried out in a second-moment turbulence modeling framework using a finite volume technique.

Their results show that, in general, the thickening of the wall boundary layer causes the non stationary motions in the cylinder wake to persist. For cylinder-to-wall gap widths \( \left( \frac{S}{D} \right) \) near that required for the suppression of vortex shedding, decreasing \( \delta/D \), the ratio of the boundary layer thickness (fig. 2.4f) resulted in a shift from near-to far wake shear layer coupling. This shift from the near -wake to the far wake results in a rise in the shedding frequency and a drop in the time averaged drag and lift on the cylinder. In addition, the pressure distribution along the lower wall changes significantly due to the increased size of the cylinder wake.

![Schematic of geometry from Straatman and Martinuzzi study (2001)](image)

Fig. 2.4d Schematic of geometry from Straatman and Martinuzzi study (2001)
Fig 2.4e Growth of Boundary layer as a function of Streamwise position of a channel with no cylinder (a) and Profiles for four different Boundary layer thicknesses.

Fig 2.4f Influence of Strouhal No., $St$ by gap width $S/D$ and boundary Layer thickness
2.5 Vortex Induced Vibrations: The lock-in phenomenon

It has been shown so far, the inter-dependency of both the body geometry and Reynolds number in determining the Strouhal number. The frequency of the shedding, $f$, is also that of the alternating forces acting transversely to the flow around the body, while the forces acting in the direction of flow have a frequency of $2f$. It is of importance that this only describes the principal oscillating forces.

Fig. 2.4g(a) & 2.4g(b) Plots of Vorticity showing the wake activity changes with boundary layer thickness - Straatman and Martinuzzi study (2001)
Consider a case of a structure that is elastically mounted, the periodic force exerted by the process of vortex shedding gives rise to oscillations. This will also influence the flow pattern and a complex interaction takes place. Invariably, separation of flow develops around the structure. The separated fluid then exerts forces on the structure; a pressure force at the leading edge and suction on the trailing edge. This alternating force of pressure and suction creates vortices in the wake region, and inevitably causing structural deformation of the deck section. According to Larsen and Walter (1996), the shedding of vortices balances the change of momentum along the entire body surface.

The shed vortices which are convected downwind by viscous diffusion and the local mean free stream wind speed will diffuse but while interacting to form large-scale coherent structures. The structural response is largely controlled by the frequency at which the vortices are shed. As long as the frequency of the vortex shedding is not close to the natural frequency of the structural member, it acts as if rigidly fixed. When and if the vortex induced and the naturally induced frequencies are the same, the resulting condition is known as the lock-in phenomena. According to Simiu and Scanlan (1986), at this time the structural member will oscillate with increased amplitude but rarely exceeding half of the across wind dimension of the body. The lock-in condition is shown in figure 2.5a.
In order to correctly evaluate conventional wind tunnel test results, Reynolds number effects on the steady and unsteady aerodynamic force coefficients must be evaluated. In general wind tunnel tests are typically in the low Reynolds number region. As has been shown earlier, the flow around bluff bodies is characterized by flow separations that form vortex streets in its wake. The Reynolds number plays a significant role in this process because flow separations are often Reynolds number dependent, even when bodies have sharp edges. This dependency is judged to arise has a result of the influence of the

**Fig. 2.5a Qualitative trend of Vortex shedding with Wind Velocity during lock-in. From Simiu & Scanlan(1986)**
boundary layer on the flow field around a body. In particular, the location of the laminar/turbulent transition point in the boundary layer or in the separated shear layer. It is understood that both the state of the boundary layer and the location of transition are responsible for the formation, length and for the topological structure of the flow. Also, on many bluff bodies, the Reynolds number influences the angle at which the shear layer separates from the body, particularly on upstream corners.

Schewe & Larsen (1998) and Schewe (2001) studied by experiments carried out over a wide range of Reynolds numbers \((10^4 < \text{Re} < 10^7)\) on 2-Dimensional sections in a high pressure wind tunnel the influence of large Reynolds number. Three bodies of importance were studied; a circular cylinder, a sharp edged trapezoidal shaped long span bridge deck section from the Great Belt East Bridge, and a 27% thick airfoil section mounted at a high angle of attack. The circular cylinder is a classical bluff body which is known to exhibit strong Reynolds number effects (fig. 2.6a-1) while the trapezoidal shaped bridge box girder section with a sharp edged section that can be considered as less bluff does not display any Reynolds number effect.

The characteristics of the flow past the cylinder are illustrated to demonstrate the effect of the laminar/turbulent transition for a bluff body; the Reynolds number effects are clear and evident both above and below the critical flow regime as shown in figure 2.6a-1. This transition from laminar to turbulent flow shows characteristics that are known to be generally valid, since they occur in flow over bodies having other cross-sections. The key role of this transition is that it wanders upstream with increasing Reynolds number. The
relationship between the static force coefficients and the Reynolds number for the cylinder are dependent on whether the flow is subcritical, supercritical or in transition. Figure 2.6a-1a shows the relationship between the drag coefficient and Reynolds number for a circular cylinder and the transition from subcritical (A) to supercritical (B). Comparing this to figure 2.6a-3, it is evident that similarity exists whether the flow is subcritical or supercritical.

The lift response is far more complex as it depends on whether the flow is subcritical or supercritical. In general, the lift coefficient increases gradually with the Reynolds number up to the laminar/turbulent transition where the flow is subcritical. According to Koide et al. (2004), when the flow becomes supercritical, the cylinder then oscillates due to the alternating lift force because of the periodic Karman vortex shedding, this oscillation influences the movement of the separation point and enhances the lift coefficient.

![Diagram showing the relationship between drag coefficient and Reynolds number for a circular cylinder.](image)

*Fig. 2.6a-1 Classical example for strong Reynolds number effect. - Schewe (2001)*
Fig. 2.6a-1a Drag Coefficient vs. Reynolds for circular Cylinder showing transition from sub critical (A) to supercritical (B).-Schewe (2001)

Fig. 2.6a-2 Sketched flow regime around trapezoidal box girder bridge.-Schewe (2001)
As for the trapezoidal box girder (fig. 2.6a-2), within the Reynolds number range of up to about $4 \times 10^4$, the drag coefficient decreases with increasing Reynolds number. Above this range the drag coefficient remains constant as the Reynolds number increases. The moment coefficient on the other hand shows few Reynolds number effects, especially in the high Reynolds number range. As for the lift coefficient (fig 2.6-a3), Reynolds number effect is not so easily defined; the pattern that emerges is non-linear with half sine wave form.

![Graph showing drag and lift coefficients](image)

**Fig. 2.6a-3 Reynolds No. effect on Drag & Lift coefficient on Trapezoidal box girder**

*bridge.-Larsen (1998)*
For all models, the distinguishing and classifying criteria of the flow field is the location of transition: Subcritical, Supercritical and Transcritical (fig. 2.6a-4). Subcritical flow reportedly exists when the flow regime is laminar upon separation, and transition occurs in the separated shear layer. The width of the wake is large, which for the circular cylinder and bridge section, leads to high drag and a low Strouhal number. The supercritical state is characterized by the formation of separation bubbles, made possible by the location of transition close to the body and as a result of this bubble formation and the now turbulent boundary layer, the separation is forced into a longer path length, a drastic reduction in the wake width and a minimum value of drag coefficient $C_d$ while the Strouhal number is maximum for the cylinder and bridge deck section. In the Transcritical state, transition occurs before separation, for the bridge section, it was concluded that the more forward location of the transition has no significant influence on the width of the wake, or on the value of either the Strouhal number or $C_d$. It then follows that the Reynolds number effect are caused by changes in the topological structure of the wake and that the location of laminar/turbulent transition plays a significant role in these structural changes, with slender bodies having sharp edged sections suffering pronounced Reynolds number effect.

It is worth noting that the Great Belt East Bridge model does not have any parapets or such attachments on it and it would be interesting to know if their observations and conclusions stated would be influenced by such details.
Matsuda, Tokushige et al. (1999) also performed wind tunnel tests and numerical simulations on 1:10 scale models of a bridge section to study the Reynolds number effects, using both steady and unsteady forces. Their conclusion suggests that flutter wind speeds obtained from unsteady aerodynamic force coefficients are generally the same or slightly higher than flutter speeds at lower Reynolds number, and Reynolds number effects on the unsteady aerodynamic forces coefficients were observed for the bridge sections where the steady aerodynamic forces and Strouhal number were both clearly influenced by the Reynolds number.
2.7 Effect of Shapes on Flows

On a typical bridge, there are attachments that are permanent and some temporary, such as parapets and barriers. Also meteorological changes such as snow and ice formation change the shapes of these objects, contributing to the aerodynamic characteristics and loadings on the structure.

Zdravkovich (1977) provided an extensive study of two similar circular cylinders in line with each other and identified five different flow patterns in relationship of the spacing between the cylinders. The relevance of this study is that such elements are often found on bridge deck sections in form of posts intermittently installed in either the longitudinal or transverse direction or quite often in both. He concluded that:

- When the gap between the cylinders is small, the shear layer that separate from the first cylinder shoots beyond the down stream cylinder. The observed characteristics of the ensuing wake are similar to those for a single cylinder.
- The generated vorticity from the upstream cylinder creates both the body and vortex shedding.
- As the flow develops, the shear layer off the upstream cylinder reattaches alternately on the sides of the second cylinder at the same period with vortex shedding.
- An irregular shedding and loading develops on the second cylinder from a quasi-steady reattachment as the gap between the cylinders increases.
• The boundary layer formation is disrupted and the vortex street becomes intermittent when the second cylinder is located close to the end of the recirculation area of the upstream cylinder.

• When the gap between the cylinders is increased, the vortex shedding in their wake is now distinct with no synchronization whatsoever.

Havel et al (2001) studied the flow pattern of two similar prismatic square cross-sectioned (fig 2.7) body arranged in parallel and placed with the face parallel to the flow to investigate the interactive dynamic properties of the flow with the bodies, such as vortex shedding and loading characteristics as a function of obstacle separation. The relevance of this study is not only in the shape of the cylinder but the observed flow pattern resulting in the variation of the distance between the obstructions, this is typical of parapet spacing between opposite sides of carriageway on a bridge deck. From this study, different types of shedding were identified for which the primary differences are as a result of the interaction between the separated shear layer from the first body and the inline face of the second body. For a 3-D configuration of similar study where $1.5 < s/d < 2.3$,
Martinuzzi & Havel (2000) identified a lock-in phenomenon. In this case, the shear layer of the upstream body impinges on and interferes with the leading face of the downstream body from it and creating a strong vortex roll in the space between them which in turn
resonates at a constant Strouhal number. It is further observed that for larger obstacle spacing, a single shedding frequency is observed in the gap and the downstream wake. The Strouhal number here increases with \( s/d \), from that of the 2-Dimensional (figure 2.7ba) case to that of an isolated obstacle.

While the study of the cylinder with right-angled corners is useful, many bluff bodies have other types of free-end shapes that are equally relevant to bridge deck section study. From an Aeroelastic/aerodynamic point of view, it is desirable to know the contributions of different free end shapes such as posts that hold the railings, on the wake characteristic of a bluff body like the bridge deck section.

Park & Lee (2004) studied the flow structure behind 4 different finite cylinders embedded in an atmospheric boundary layer (fig. 2.7). The velocity time series were analyzed to investigate the effect of the free-end shape on the vortex shedding frequency. Past studies on 2-D circular cylinder, have shown that large scale vortices are shed on both sides of the cylinder, and are almost always uniform along the cylinder’s spanwise direction. Although the flow can be nominally assumed to be 2-Dimensional, in reality there is a large amount of three dimensionality to the flow. The vortices shed to form the Karman Street are not necessarily in phase over the span wise length, and also, at certain Reynolds numbers, a phenomenon of oblique vortex shedding can arise. As the Karman vortices are convected away from the body, 3-Dimensional effects such as vortex roll-up or vortex stretching arise. The process of vortex shedding on the 3-Dimensional cylinder is somewhat different; the process is more influenced as a result of the downwash.
separated flow from the cylinder's free end. This separated shear flow becomes dominant and it affects the structure of the wake behind three-dimensional cylinders with free end shapes.

Fig 2.7c Free-end shapes tested by Park et al. (2004)
2.8 Numerical Simulations.

In this section, the application and concepts related to the computer modeling of the flow around bridges is discussed. Detailed procedures including grid generation, turbulence models, and boundary conditions are left for later chapters. Using the Finite Difference Method (FDM), (Fujiwara et al. 1993) were the first to attempt numerical simulation on bridge deck section. The authors carried out 2-Dimensional numerical simulations of a flow field around an elastically supported edge beam cross-section with fairings, using the method of coordinate transformation. The Navier-Stokes solutions presented were for $Re$ in the range of $2100 - 4000$ based on wind tunnel experiments.
Onset wind speed predictions were in good agreement for cross-sections for which vortex induced oscillations had been experimentally observed. For cross-sections for which vortex induced oscillations were not observed, the numerical simulations yielded oscillations with large amplitudes. It was concluded that agreement between numerical simulations with experimental ones thus varied with the shapes of the body and possibly different computational method will be needed for different body shapes.

Mendes and Branco (1995) followed with a CFD simulation of a rectangular section with a side ratio (B/D=4) at a Reynolds number of 500. Using an Arbitrary Lagrangian-Eulerian (ALE) formulation in a finite element code for the direct Navier-Stokes equation, the authors were able to obtain Scanlan’s flutter derivatives. However, this solution strategy does not employ a turbulence model.

Onyemelukwe (1993) developed a 2-Dimensional finite difference solver on a boundary fitted grid for Laminar Navier Stokes flow solutions for a variety of fixed bridge decks for Re of about $1 \times 10^5$. The studies were limited by restrictions in computer CPU and memory and static force coefficients could not even be computed.

Bienkiewicz and Kutz (1993) introduced the mathematical foundation of the Discrete Vortex Method (DVM) while addressing computer modeling issues associated with application of the method for bluff body applications. In the Discrete Vortex Method, the is represented by an array of finite vortices which mutually interact as the flow develops. The velocity at each vortex is the cumulative sum of the velocities induced by all other
vortices. In the DVM method, for a 2-D incompressible flow the conservation of mass is

given as:

\[ \nabla \cdot u = 0 \quad (1.22) \]

\[ \frac{\partial u}{\partial t} + u \cdot \nabla u = -\nabla p + \frac{1}{R_e} \Delta u \quad (1.23) \]

where \( u = (u,v) \), \( p \), and \( t \) are the non-dimensional velocity, pressure and time, and \( Re \) is the

Reynolds number. The vorticity transport equation is obtained by taking curl of equation

\( (1.18) \) thus:

\[ \frac{\partial \omega}{\partial t} + u \cdot \nabla \omega = \frac{1}{R_e} \Delta \omega \quad (1.24) \]

where the vorticity is given by

\[ \omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \quad (1.25) \]

From equation 1.19 two contributing mechanisms to vorticity transport are identified,

namely, advection (left side) and diffusion (right side). Vortex methods treat these two

mechanisms in two fractional steps, where:

\[ \frac{D \omega}{Dt} = 0 \quad (1.26) \]

and

\[ \frac{\partial \omega}{\partial t} = \frac{1}{R_e} \Delta \omega \quad (1.27) \]
The attraction to the DVM is that it is grid free, thereby removing difficult and time-consuming process of meshing. Walther (1994), applied the DVM method for solution of 2-Dimensional vorticity equation for the flow around a cross-section at \( Re=10^5 \). Satisfactory agreement were obtained between simulations and experiment. Larsen (1998) also used the DVM to simulate flow past the girder cross-section of the Great Belt East Bridge (GBEB) good agreements were obtained with the static parameters of the aerodynamic characteristics. Morgenthal and Mcrobie (2002) also applied the DVM to rectangular cylinders as shown in figure 2.8a as well as the GBEB, both reported good agreement with wind tunnel tests.

![Fig. 2.8a Vortex Shedding from Rectangular Cylinders using the DVM (Morgenthal & Mcrobie-2002)](image_url)
Taylor and Vezza (2001) also applied a modified version of the discrete vortex method, in this case the basis of the method is the discretization of the vorticity field, rather than the velocity field, into a series of vortex particles that are free to move in the flow field that the particles collectively induced. Applications on a suspension bridge deck section shows good agreement with wind tunnel tests.

Folch, et al (2003) performed numerical analysis using CFD to predict the lift, drag and moment coefficients of the Great Belt Bridge. This work is interesting in two ways; firstly, the wind tunnel test on which the numerical simulation was compared was done on two model scales; a 1:80 and 1:300 scales. The drag coefficients on both scaled models were fairly similar, however, the lift and moment coefficient vary remarkably even though the details of both models were similar. Secondly, the simulation was done using both laminar and turbulent flow respectively. Turbulent solution was based on the Spalart-Almaras model (see chapter 3), while there seems to be good correlations between the numerical simulations with the wind tunnel test, it is interesting that using either laminar or turbulent flow has very little influence on the solution.

Tubelin and Gibert (2002) demonstrated that computational fluid dynamics (CFD) can be used to obtain indicial lift responses and to derive admittance functions which are used to improve buffeting analysis and for calculating the frequency dependent wind induced forces associated with a gust of variable velocity on bluff bodies similar to long span
bridge sections which initiate detached flows, their studies suggest the normal use of the classical airfoil functions for this type of phenomenon may not be appropriate, also for bluff bodies, indicial and admittance functions for lift are dependent on the evolution of the separated flow around the section at the early stages.

The effect of edge details has also received attention: e.g. Robertson, Sherwin and Bearman (2003). Robertson, Sherwin, Bearman and Li (2003) studied both the rotational instabilities of rectangular and semi circular leading edge sections as well as the rotational and galloping response of rectangular bodies (See fig. 2.8a-d). Their studies, although using simple geometries, demonstrated that in the case of galloping, the leading edge is of primary importance to the critical reduced velocity of the onset of rotational galloping. In these studies, a rectangular section was shown to have a negative variation of moment coefficient with increasing angle of attack, and from the quasi-steady theory implies the body will undergo rotational galloping at a sufficiently high reduced velocity. Sections with a semi-circular leading edge have a positive moment coefficient and therefore by the quasi-steady theory do not have the potential to gallop although it may suffer divergent instability. It should be noted that the flow in this study is laminar since the Reynolds number was rather low.
Fig. 2.8b Geometry used by Robertson et al. (2003) in the investigation of Rotational Galloping instability (a) Rectangular leading edge (b) Semi circular leading edge.

Fig. 2.8c Vorticity and Pressure field around galloping rectangular section. From Robertson et al. (2003)

Fig. 2.8d Vorticity and Pressure field around galloping semi-circular leading edge section. From Robertson et al. (2003)

Bruno et al. (2001) used both the standard and the modified $k - \varepsilon$ turbulence models (see chapter 3) to study the aerodynamic behavior of equipped bridge deck section. The study like other attempts at CFD application to bridge deck section study was 2-dimensional. Based on the studies, the authors concluded that the $k - \varepsilon$ turbulence
model with standard wall functions is unable to reproduce the separation that often occur in flow around bluff bodies, and that minor changes made to the cross-sectional geometry by equipments or fairings have an importance influence not only in the neighborhood of their location, but also on the aerodynamic behavior of the section as a whole. The study however, did not examine the effects of details on deck response to vortex-shedding, a subject that the authors suggested or hope for later investigation which is part of the effort of the current work by this author.

Larsen and Walther (1998) did numerical studies based on the discrete vortex method (DVM) on five generic bridge deck sections. The same sections had previously been studied experimentally by Scanlan and Tomko (1972) who reported aerodynamic flutter derivatives for use in flutter analysis. This work like many others before it and after it that deals with edge details was 2-dimensional. The five sections are shown in fig. 2.9 below. The authors reported some reasonable correlations between some aspects of the flutter derivatives coefficient with those obtained experimentally. Static force coefficients were not compared with experimental wind tunnel tests but rather numerically with various deck sections as shown in figure 2.9.
Steady state load coefficients and flow field at time $tU/B = 10$

<table>
<thead>
<tr>
<th>Section</th>
<th>$C_D$</th>
<th>$C_L^{rms}$</th>
<th>$St$</th>
</tr>
</thead>
<tbody>
<tr>
<td>G1</td>
<td>0.08</td>
<td>0.07</td>
<td>0.17</td>
</tr>
<tr>
<td>G2</td>
<td>0.08</td>
<td>0.08</td>
<td>0.17</td>
</tr>
<tr>
<td>G3</td>
<td>0.10</td>
<td>0.08</td>
<td>0.10</td>
</tr>
<tr>
<td>G4</td>
<td>0.08</td>
<td>0.12</td>
<td>0.17</td>
</tr>
<tr>
<td>G5</td>
<td>0.27</td>
<td>0.33</td>
<td>0.11</td>
</tr>
</tbody>
</table>

**Fig. 2.9** Geometry and flow field of generic bridge girder sections from Larsen & Walther (1998). Where $C_D$, $C_L^{rms}$, $St$ are Drag, Lift (Root Mean Square) coefficient & Strouhal number respectively.
As part of the wind tunnel and aerodynamic studies of the Aki-nada Ohashi Bridge in Japan (fig 2.9b), Honda et al (1998) reported on the influence and sensitivity of the modification of additional members (parapet, rail for maintenance cargo, etc) on the response of the deck section. In this particular study, minor changes of parapets succeeded in increasing the stability of the deck after the completion of the bridge thereby underscoring the importance and the rationale for this type of work numerically.

![Fig. 2.9b Aki-nada Ohashi Bridge Deck Section with modified parapet (b) as part of edge details to control bridge stability-Honda et al (1998).](image)

Selvam, Govindaswamy & Bosch (2002) studied the issues involved in the computation of flow around bridges and to compute the critical velocity for flutter in a direct way using a moving grid. The moving grid methodology is commonly used to solve the problem encountered in fluid-structure interaction modeling, this is because the equation of motion of the structure and the fluid must be solved simultaneously. Since the
structural equations are formulated in the Lagrangian coordinate system while the fluid equations are in the Eulerian coordinate system, hence a moving grid is needed at each time step for the fluid portion in order to solve the equations. Several procedures have been proposed to solve the problem, such as the arbitrary Lagrangian-Eulerian (ALE) formulation proposed by Nomura and Hughes (1992), co-rotational approach proposed by Murakami and Mochida (1995) and the dynamic meshes by De Sampio et al. (1992). In the ALE, grid can be moved as a whole with constant velocity for each node or with different velocity for each node and in some regions no movement at all. Thus it is the most computationally easy to apply and widely used for fluid-structure analysis.
2.9 Summary.

This Chapter reviewed the history and development of bridge aerodynamics spurred by the collapse of the Tacoma Narrows Bridge. Aeroelastic phenomena that are pertinent to long span bridge deck sections, flow behavior of isolated elements and apparatus that may be found on bridge deck sections and current efforts at numerical simulations of deck sections are also discussed. The main findings may be summarized as follows:

- There is a clear understanding on the physics and methodology of the aeroelastic phenomenon of Long Span Bridge sections.
- Isolated flow study of different sections and shapes has helped to define various type of vortex shedding and vortex induced oscillations.
- Computer simulation of long span deck sections is increasingly becoming an essential tool for better understanding and the design of long span bridges as complementary to wind tunnel test.
- Numerical simulations of bridge deck sections to-date have been 2-dimensional often without the parapets, barriers or railings as part of the model.
- The influence of these elements and edge details on long span bridge sections has not being studied numerically in 3-dimensional fashion is not well understood.

It is further concluded that due to limited geometry studied by various researchers, and the fact that the models are 2-Dimensional, the influence of parapets and edge details on the Aeroelastic/aerodynamic properties are not well understood and therefore need further studying. It is proposed that a 3-Dimensional model is essential and indeed required to
better understand their effects of these elements on the flow patterns and characteristics of long span bridges.
Turbulence Modeling & Solution Strategies.

3.0 Introduction

In general, most flows encountered in the practice of fluid mechanics and computational fluid dynamics are often turbulent and characterized by unsteadiness, highly 3-Dimensional and contain a great deal of vorticity, which can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to the viscous forces, and is characterized by a high enough Reynolds Number.

To enable the effects of turbulence to be predicted, a large amount of CFD research has concentrated on methods which make use of turbulence models. Turbulence models have been specifically developed to account for the effects of turbulence without recourse to a prohibitively expensive fine mesh. Most turbulence models are statistical turbulence models as described in this chapter. If one considers time scales that are larger than that of turbulent fluctuations, it can be surmised that turbulent flow display average characteristics. For instance a velocity component can be divided into an average component and a time dependent component. In general, turbulent models work through modifying the original and unsteady Navier-Stokes equations by introducing averaged and fluctuating properties to produce the Reynolds Averaged Navier Stokes (RANS)
equations. These derived equations represent the mean flow quantities only, while modeling turbulence effects without a need for the full space and time resolution of the turbulence effects.

Turbulence models of this nature are termed Statistical Turbulence Models, as a result of the statistical averaging procedure used in generating the equations.

In the past, the primary means of studying turbulent flows were experimental. However, in the last few decades several numerical methods have been developed for solution of turbulent flows and more specifically the unsteady Navier-Stokes equations in their conservation form. This growth coincidentally is directly related to similar geometric growth in computer power.

Bardina et al (1980), in common with many writers, suggested a categorization of these methods based on the following criteria:

- Methods that use correlations such as ones that give the friction factor as a function of the Reynolds number. This is normally of limited use and only for flows that have few parameters.
- Methods using integral equations which are essentially derived from the equations of motion integrated over some coordinate direction (or time).
- Methods based on averaging the equations of motion over time. These are often referred to as the one-point closure and lead to a set of partial differential equations called the Reynolds Averaged Navier-Stokes or RANS.
- Two point closures that use equations for the correlation of the velocity components at two spatial points and more often the Fourier transformation of these equations.
- Large Eddy Simulation (LES) that solves for the largest scale motions while approximating the small scale motions
- Direct Numerical Simulation (DNS) in which the Navier-Stokes equations are solved for all aspects of the turbulent flows.

The categorizations suggested by Bardina et al (1980) are very useful in understanding and implementing the most appropriate solution method for a given problem. The discussion in this chapter is not so much about the understanding of the mathematical theory of a particular turbulent model, but rather to provide a bridge designer wishing to use a typical CFD solver to obtain preliminary aerodynamic characteristics, enough information about the turbulent models and what works for this type of study.

In order to move this research forward, it was necessary to conduct a parametric study on the turbulent models so as to determine the most effective models for external flows of the type discussed in this research. The turbulent models discussed here are universal and are not peculiar or particular to any CFD solver; while their implementations may differ slightly from code to code, they are all available almost without exception in all the commercially available solvers, therefore no attempt is made to endorse or reference given to a particular commercial code.
3.1 Navier-Stokes Equations.

In principle, the Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of length and time scales and would generally involve length scales much smaller than the smallest finite volume mesh which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows would require computing powers that are at the moment not practical.

As described above, turbulence models seek to solve a set of modified transport equations by introducing averaged and fluctuating components. For example, the velocity $U$ may be divided into an average component $\overline{U}$, and a time varying component, $u$. Thus the velocity may be written as:

$$U = \overline{U} + u$$

And the averaged component is given by:

$$\overline{U} = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} U dt$$

Where $\Delta t$ is the time scale that is large relative to the turbulent fluctuations, but small relative to the time scale to which the equations are solved.

Substituting the time averaged quantities in the original transport equations results in the Reynolds-averaged equations given below. In these equations, the bar has been dropped for the time averaged quantities, except for products of fluctuating quantities.
The basic Navier-Stokes instantaneous equations of continuity, momentum and energy conservation as implemented in the CFD code Ansys-CFX can be written as:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{3.1}
\]

\[
\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = \nabla \cdot (- \rho \mathbf{\delta} + \mu (\nabla \mathbf{U} + (\nabla \mathbf{U})^T)) + S_M \tag{3.2}
\]

\[
\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U} h_{tot}) = \nabla \cdot (\lambda \nabla T) + S_E \tag{3.3}
\]

where:

- \( \mathbf{U} \) = vector of velocity
- \( T \) = static temperature
- \( S_M \) = mass source
- \( S_E \) = energy source
- \( \nabla \) = vector operator, such that \( \nabla = \left[ \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right] \)
- \( \nabla \cdot \) = vector operator, such that for a vector function \( \mathbf{U}(x,y,z) \) the divergence of \( \mathbf{U} \) is defined by \( \nabla \cdot \mathbf{U} = \frac{\partial U_x}{\partial x} + \frac{\partial U_y}{\partial y} + \frac{\partial U_z}{\partial z} \)
- \( \otimes \) = a tensor product, an operation which is a multiplication of one vector by another
- \( \mathbf{\delta} \) = the identity matrix
- \( \lambda \) = thermal conductivity
- \( h_{tot} \) = specific total enthalpy
3.2 Direct Numerical Simulation.

The most accurate methodology for turbulent simulations is to solve the Navier-Stokes equations without averaging using a numerical discretization where the errors associated with it are either controlled and or reasonably estimated. When the direct numerical simulation is applied, the computational domain must be large enough and the computational grid dense, otherwise all the futures of the turbulence will not be properly described. The proper length of the domain is set by the so called integral length scale, defined as the distance after which the self correlation of the velocity components vanishes. In a turbulent flow, the ratio of the integral scale to the micro-scale is proportional to $Re^{\frac{1}{4}}$, so then the number of nodes in a DNS simulation increases as $(Re^{\frac{1}{4}})^3$, i.e. $Re^{\frac{3}{4}}$. The same requirement holds for the time step, so the computing power increases as $(Re^{\frac{1}{4}})^4$, i.e. $Re^3$. This is why DNS is confined to low $Re$ flows.

The number of grid points is set by the Reynolds number, as the Reynolds number increases, the ratio of the integral length scale to the dissipation scales in the flow grows as well, because the dissipation scale decreases the (smallest eddies become smaller) in the flow field. Therefore, the grid spacing or cell sizes must be able to capture all these small scale motion. For these reasons, the direct numerical simulation requires so much computer capacity that it is not adaptable for normal engineering type flows.

The major role the DNS fills is as a research tool with applications such as in understanding the effects of compressibility on turbulence, controlling and or reducing drag on a solid surface and simulation of aerodynamic noise.
3.3 Large Eddy Simulation (LES) Model.

Large Eddy Simulation (LES) is about filtering of the equations of movement and decomposition of the flow variables into large scale (resolved) and a small scale (unresolved) parts. Filtering is essentially a manipulation of the exact Navier-Stokes equations to remove only the eddies that are smaller than the size of the filter, which is normally taken as the size of the mesh. Similar to the Reynolds averaging, this filtering process creates additional unknown terms which in turn must be modeled in order to achieve closure. The statistics of the mean flow quantities that are generally of engineering interest are obtained during the transient simulation. The advantage that the LES offers over the RANS model(s) is that, by modeling less of the turbulence, there is a reduction in the error induced by the turbulence model.

The filtered variable (say velocity) is defined by

$$\overline{u}_i(x) = \int G(x,x')u_i(x')dx$$

where $G(x,x')$ is the Gaussian filter function that determines the scale of the resolved eddies. As indicated earlier, the length scale is normally the size of the mesh $\Delta$, eddies of size larger than $\Delta$ are large eddies while those smaller than $\Delta$ are small eddies, those that need to be modeled. After performing the volume averaging and disregarding the density fluctuations, the filtered Navier-Stokes equations become:

$$\frac{\partial}{\partial t}(\rho \overline{u}_i) + \frac{\partial}{\partial x_j}(\rho \overline{u}_i \overline{u}_j) = \frac{\partial}{\partial x_j}\left( \mu \frac{\partial \overline{u}_i}{\partial x_j} \right) - \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j}$$

(3.4)

Since the continuity equation is linear, the filtering does not change it:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho \overline{u}_i) = 0$$

(3.5)
It should be noted that in equation 3.4 the quantity $u_i \mu_j \neq \mu_{i j}$, and that the quantity $u_i \mu_j$ is not easily computed. The modeling approximation between both sides of the inequality in the context of the LES is referred to as the subgrid-scale Reynolds stress, defined as:

$$\tau_y = -\rho \left( u_i \mu_j - \mu \frac{\partial \overline{u_i}}{\partial x_j} \right)$$

The models used in approximating the SGS Reynolds stress are referred to as the subgrid-scale models. Most of these models are of the eddy viscosity type in this form:

$$\tau_y = -\frac{1}{3} \mu \delta_y = \mu \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) = 2 \mu \overline{S_y}$$

where $\mu$ is the eddy viscosity and $\overline{S_y}$ is the strain rate of the large scale or resolved field.

Further studies on the LES and the solutions of the subgrid scale model are found in the works of Smagorinsky (1963), Lilly (1966) and Yakhot et al (1989).

3.4 Summary of the Reynolds Averaged Navier Stokes (RANS) Models

Simulations of the RANS equation and LES reduces the computational effort compared to a Direct Numerical Simulation and is generally adopted for practical engineering calculations. However, the averaging procedure introduces additional unknown terms containing products of the fluctuating quantities, which act like additional stresses in the fluid. These terms, called turbulent or Reynolds stresses, are difficult to determine and become further unknowns. The Reynolds (turbulent) stresses need to be modeled by additional equations of known quantities in order to achieve “closure”. By closure, it implies that there is a sufficient number of equations for all the unknowns,
Turbulence Modeling and Solution Strategies

Chapter

including the Reynolds-Stress tensor resulting from the averaging procedure. The equations used to close the system define the type of turbulence model. The following are examples of the commonly used turbulence models ranked by degree of complexity.

• **Eddy Viscosity Turbulence Models.** The eddy viscosity model assumes that the Reynolds stresses can be related to the mean velocity gradients and eddy (turbulent) by the viscosity gradient diffusion hypothesis. The Reynolds stresses are assumed to be proportional to mean velocity gradients and eddy (turbulent) viscosity. This defines an eddy viscosity model.

• **The Zero Equation Model.** The very simple eddy viscosity models compute a global value for $\mu_t$ from the mean velocity and a geometric length scale using an empirical formula. Since no additional transport equations are solved, these models are termed *zero equation.* The turbulence viscosity is modeled as the product of a turbulence velocity scale, $U_t$, and a turbulence length scale, $l_t$, as proposed by Prandtl and Kolmogorov,

$$\mu_t = \rho f_{\mu} U_t l_t \quad (3.6)$$

where $f_{\mu}$ is proportionality constant.

The length scale for example can be derived using equation 3.7 among several available methods:

$$l_t = \left( \frac{1}{V_D^3} \right)^{1/7} \quad (3.7)$$

where $V_D$ is the fluid Domain volume.
• **One equation Models.** One equation turbulence models as the name suggests, solve just one turbulence transport equation. The original one equation model is the Prandtl one-equation model, others are the Baldwin-Barth model and the Spalart-Almaras model, which will be further discussed later. Typically, the equation only solves the turbulent kinetic energy or turbulent viscosity. One advantage of this type of model is that additional assumptions can be avoided. However, the length scale is still defined through algebraic equation.

• **Two equation Models.** The two equation models are more sophisticated than the zero or one equation models. Here, both the velocity and length scale are solved using separate transport equations hence the term *two equation*. The \( k - \varepsilon \) and the \( k - \omega \) two equation models use the gradient transport hypothesis to relate the Reynolds stresses to the mean velocity and the turbulent viscosity. In the \( k - \varepsilon \) model, \( k \) is the turbulent kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of \( (L^2T^{-2}) \), \( \varepsilon \) is the rate of dissipation of turbulence energy and has dimensions of \( (L^2T^{-3}) \). The \( k - \omega \) formulation is still more advantageous since the model does not involve complex non-linear damping functions required in the \( k - \varepsilon \) model (CFX-Theory manual) and is therefore more accurate and more robust. The \( k - \omega \) model assumes that the turbulent viscosity is linked to the turbulence kinetic energy and turbulent frequency via the relation:

\[
\mu_t = \rho \frac{k}{\omega} \tag{3.8}
\]
• The shear Stress Transport (SST) $k-\omega$ Based Model. The $k-\omega$ based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients (Menter, 1994). The SST model combines the advantages of both the $k-\omega$ and the $k-\varepsilon$.

• Reynolds Stress Turbulence Transport Models. These models are based on transport equations for all six components of the Reynolds Stress tensor and the dissipation rate. These models do not use the eddy viscosity hypothesis, but solve an equation for the transport of each Reynolds stresses in the fluid. The Reynolds stress model transport equations are solved for the individual stress components. Somewhat simplified algebraic Reynolds stress models solve algebraic equations for the Reynolds stresses. The exact production term and the inherent modeling of stress anisotropies theoretically make Reynolds Stress models more suited to complex flows; however, practice shows they are often not superior to two equation models.

3.5 The $k-\varepsilon$ Model

Of the two equation models, the most widely used model has being the $k-\varepsilon$ model. First proposed by Launder and Spalding (1974), the model has since been modified and currently exists in a number of variant forms; including:

• The standard $k-\varepsilon$

• The RNG (Renormalization Group theory) $k-\varepsilon$
Turbulence Modeling and Solution Strategies
Chapter 3

- The Realizable $k-\varepsilon$ Model

The $k-\varepsilon$ model introduces two new variables into the RANS equations, the continuity equation is written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0$$  \hspace{1cm} (3.9)

and the momentum equation becomes:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) - \nabla \cdot (\mu_{eff} \nabla U) = \nabla p^l + \nabla \cdot (\mu_{eff} \nabla U)^T + B$$  \hspace{1cm} (3.10)

where $B$ is the sum of body forces, $\mu_{eff}$ is the effective viscosity accounting for turbulence, and $p^l$ is the modified pressure given by $p^l = p + \frac{2}{3} \rho k$. The $k-\varepsilon$ models based on the eddy viscosity model. The eddy viscosity hypothesis suggests that turbulence consists of small eddies which are forming and dissipating, and in which the Reynolds stresses are assumed to be proportional to mean velocity gradients. It is also assumed that the Reynolds stresses can be related to the mean velocity gradients and eddy (turbulent) viscosity by the gradient diffusion hypothesis. In equation 3.10, $\mu_{eff}$ is the effective viscosity defined by,

$$\mu_{eff} = \mu + \mu_t$$

where $\mu_t$ is the turbulence viscosity. The $k-\varepsilon$ assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation,

$$\mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon}$$
where $C_\mu$ is a standard value constant typically given as 0.09. The values of $k$ and $\varepsilon$
come directly from the differential transport equations for the turbulent kinetic energy
and turbulence dissipation rate:

$$\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \vec{U} k) = \nabla \cdot \left[ \left( \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \rho \varepsilon$$ (3.11)

$$\frac{\partial (\rho \varepsilon)}{\partial t} - \nabla \cdot (\rho \vec{U} \varepsilon) = \nabla \cdot \left[ \left( \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon)$$ (3.12)

where $C_{\varepsilon 1}$ is Reynolds stress model constant given as 1.45, $C_{\varepsilon 2}$ is Reynolds stress model
constant given as 1.9, $\sigma_k$ is turbulence model constant for the $k$ equation given as 1.0
and $\sigma_\varepsilon$ is $k - \varepsilon$ turbulence model constant given as 1.3. $P_k$ is the turbulence production
due to viscous and buoyancy forces, which is modeled using:

$$P_k = \mu_t \nabla \vec{U} \cdot (\nabla \vec{U} + \nabla \vec{U}^T) - \frac{2}{3} \nabla \cdot \vec{U} (3 \mu_t \nabla \cdot \vec{U} - \rho k) + P_{kb}$$ (3.13)

For incompressible flows, the term $\nabla \cdot \vec{U}$ is small and the second term on the right side
of equation 3.13 does not contribute significantly to the production.

### 3.5.1 The RNG $k - \varepsilon$ Model

The RNG $k - \varepsilon$ model is based on renormalization group analysis of the Navier-Stokes
equations to account for the effects of smaller scales of motion. In the standard $k - \varepsilon$, the
eddy viscosity is determined from a single turbulence length scale, such that the
calculated diffusion is that which occurs only at the specified scale; however, in reality
all scales of motion will contribute to the turbulent diffusion. The RNG methodology
though similar to the $k-\varepsilon$ model, results in a modified form of the epsilon equation which attempts to account for the different scales of motion by changing the production term. In this scheme, the transport equations for turbulence generation and dissipation are the same as those for the standard $k-\varepsilon$ model, but model constants differ, and the constant $C_{e1}$ is replaced by the function $C_{e1RNG}$. The transport equation for turbulence dissipation becomes:

$$\frac{\partial (\rho \varepsilon)}{\partial t} \nabla \cdot (\rho \mathbf{U} \varepsilon) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_{eRNG}} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} \left( C_{e1RNG} \rho - C_{e2RNG} \rho \varepsilon \right)$$

(3.14)

where, $C_{e1RNG} = 1.42 - f_\eta$ and $f_\eta = \frac{\eta \left( 1 - \frac{\eta}{4.38} \right)}{\left( 1 + \beta_{RNG} \eta^2 \right)}$, $\eta = \frac{\sqrt{P_k}}{\rho C_{\muRNG} \varepsilon}$.

$C_{e2RNG} = \text{RNG } k-\varepsilon$ Turbulence model constant of 1.68, $C_{\muRNG} = \text{RNG } k-\varepsilon$ Turbulence model constant of 0.085, $\beta_{RNG} = \text{RNG } k-\varepsilon$ Turbulence model constant of 0.012. A comprehensive description of this model and its application to turbulence is available in Choudhury (1993).

The realizable $k-\varepsilon$ model proposed by Shih et al (1995) is a relatively recent development and differs from the standard $k-\varepsilon$ model in two important ways:

- The realizable $k-\varepsilon$ model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate, $\varepsilon$, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.
The term “realizable” means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. The transport equations for the realizable $k - \varepsilon$ model are given as:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[ \mu + \frac{\mu}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (3.15)$$

and

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[ \mu + \frac{\mu}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} + C_1 \frac{\varepsilon}{k} G + C_3 \varepsilon + S_\varepsilon \quad (3.16)$$

where $G_k$ represents the production of kinetic energy and is modeled identically as in both the standard and RNG models as $G_k = \mu S^2$ where $S$ is the modulus of the mean rate-of-strain tensor, defined as $S = \sqrt{\sum_{ij} S_{ij} S_{ij}}$, $C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right]$, $\eta = \frac{S}{\varepsilon}$.

$G_b$ is the generation of turbulence kinetic energy due to buoyancy and $Y_M$ represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. These last two terms are really not of interest or strictly applicable for usage in the context of the work done in this research; incompressible, non buoyant and low Mach number flows. Further studies on the usefulness of this variable may be found in Shih et al (1995).

One of the great advantages of the $k - \varepsilon$ model is its robustness and stability. However, Murakami (1993) suggests that the inherent impinging, separation and vortex shedding characteristic of the flowfield around bluff bodies cannot be accurately modeled and resolved with the $k - \varepsilon$ model. Murakami attributes this to the overproduction of the
turbulent kinetic energy, \( k \), when the \( k-\varepsilon \) model is applied to a flow field with impingement such as in sharp corners of a bridge deck section as shown in figure 3.4.

![Wind Tunnel Experiment](image1)

*Fig. 3.4 Overproduction of turbulent energy \( k \) by \( k-\varepsilon \) the model-Murakami (1993)*

Another well known deficiency of the \( k-\varepsilon \) model is its inability to handle low turbulent Reynolds number computations, further discussion on this problem can be found in chapter 6 of this thesis.

### 3.6 The \( k-\omega \) Model

The \( k-\omega \) model is another of the eddy viscosity models, unlike the \( k-\varepsilon \) model, it is a two equation model since it includes two extra transport equations to represent the turbulent properties of the flow, thereby accounting for history effects like convection and diffusion of turbulent energy. One of its advantages is the formulation for near wall treatment for low-Reynolds number computations. A low Reynolds \( k-\varepsilon \) model would
Turbulence Modeling and Solution Strategies

Chapter

normally require a near wall resolution of \( y^+ \leq 0.2 \) (see chapter 4 for discussion on \( y^+ \)) while a low Reynolds number \( k-\omega \) model would require at least \( y^+ \leq 2 \).

The \( k-\omega \) models assumes that the turbulence viscosity is linked to the turbulent kinetic energy and turbulent frequency by the relation:

\[
\mu_t = \rho \frac{k}{\omega}
\]

Broadly speaking, the \( k-\omega \) model exists in three forms, namely;

- The Wilcox \( k-\omega \) Model
- The Baseline (BSL) \( k-\omega \) Model
- The Shear Stress Transport (SST) \( k-\omega \) based Model

Of these three, the Shear Stress Transport model is vastly superior to the previous two; the reason will become evident later in the chapter.

The starting point is the formulation by Wilcox (1986), it solves two transport equations, one for the turbulent kinetic energy, \( k \), and another for turbulent frequency, \( \omega \). The \( k \)-equation is given as:

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \beta \rho k \omega
\]

(3.17)

While the \( \omega \) is given by:

\[
\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho U \omega) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_{\omega}} \right) \nabla \omega \right] + \frac{\omega}{k} P_k - \beta \rho \omega^2
\]

(3.18)

The model constants are given by:
\[ \beta' = 0.09 \]
\[ \alpha = \frac{5}{9} \]
\[ \beta = 0.075 \]
\[ \sigma_k = 2 \]
\[ \sigma_\omega = 2 \]

The unknown Reynolds stress tensor, \( \tau \), is calculated from:

\[ \tau = \mu \frac{25}{3} - \rho \frac{2}{3} \delta k \]  

(3.19)

Menter (1994), introduced a limiter to the production term in order to avoid the build-up of turbulent kinetic energy in stagnation regions. The limiter is given as

\[ \tilde{P}_k = \min(P_k, c_{lim} \omega) \text{ with } c_{lim} = 10 \text{ for } \omega \text{-based models.} \]

The limiter does not affect the shear layer performance of the model, but has consistently avoided the stagnation point build-up in aerodynamic simulations.

One of the main problems identified by Menter (1993) with the Wilcox model is its well known sensitivity to free stream conditions. Depending on the value specified for \( \omega \) at the inlet, a significant variation in the results of the model can be obtained. This is undesirable, and in order to solve the problem, a blending between the \( k - \omega \) model near the surface and the \( k - \varepsilon \) model was developed by Menter (1994). This blending consists of a transformation of the \( k - \varepsilon \) model to a \( k - \omega \) formulation and the subsequent addition of the corresponding equations. Therefore the Wilcox model is multiplied by a blending function \( F_1 \) and the transformed \( k - \varepsilon \) model by a function \( 1 - F_1 \). Near the surface \( F_1 \) is equal to 1 but switches over to zero inside the boundary layer. At the boundary layer edge and outside the boundary layer, the standard \( k - \varepsilon \) model is recovered.
Wilcox model:

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_k}{\sigma_k} \right) \nabla k \right] + \rho_k - \beta \rho k \omega
\]  

(3.20)

\[
\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho U \omega) = \nabla \cdot \left[ \left( \mu + \frac{\mu_k}{\sigma_{\omega}} \right) \nabla \omega \right] + \frac{\omega}{k} \rho_k - \beta \rho \omega^2
\]  

(3.21)

And the transformed \( k-\varepsilon \) model:

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_k}{\sigma_{k_3}} \right) \nabla k \right] + \rho_k - \beta \rho k \omega
\]  

(3.22)

\[
\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho U \omega) = \nabla \cdot \left[ \left( \mu + \frac{\mu_k}{\sigma_{\omega_3}} \right) \nabla \omega \right] + 2 \rho \frac{1}{\sigma_{\omega_3}} \nabla k \nabla \omega + \frac{\omega}{k} \rho_k - \beta \rho \omega^2
\]  

(3.23)

The Wilcox model equations can now be multiplied by the function \( F \), and the transformed \( k-\varepsilon \) model by a function \( 1 - F \), the corresponding \( k \) and \( \omega \) equations are added to give the BSL model:

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_k}{\sigma_{k_3}} \right) \nabla k \right] + \rho_k - \beta \rho k \omega
\]  

(3.24)

\[
\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho U \omega) = \nabla \cdot \left[ \left( \mu + \frac{\mu_k}{\sigma_{\omega_3}} \right) \nabla \omega \right] + \left( 1 - F \right) 2 \rho \frac{1}{\sigma_{\omega_3}} \nabla k \nabla \omega + \frac{\omega}{k} \rho_k - \beta \rho \omega^2
\]  

(3.25)

The coefficients of the new model are a linear combination of the corresponding coefficients of the underlying models.

Although, the BSL model combines the advantage of the Wilcox and the \( k-\varepsilon \) model, it still fails to properly predict the onset and amount of flow separation from smooth
surfaces correctly. The main reasons for this are given by Menter (1994). The major reason was that both models fail to account for the transport of the turbulent shear stress, resulting in overproduction of the eddy-viscosity.

The starting point for the development of the Shear Stress Transport Model (SST) by Menter (1993, 1994) was for the accurate modeling and prediction of aeronautical flows with strong adverse pressure gradients and separation, since the previously stated models had consistently failed in this task. The Johnson-King Model (1984) was the first formulation which allowed the accurate prediction of separated airfoil flows; unfortunately the model was not easily adaptable to 3-Dimensional Navier-Stokes codes due to its algebraic formulation.

As shown earlier, the $k-\omega$ model is substantially more improved over the $k-\varepsilon$ model in the near wall layers, and has therefore been successful for flows with moderate adverse pressure gradients, but fails for flows with pressure induced separation according to Menter (1993). The SST zonal model formulation is based on blending functions, which ensures a proper selection of the $k-\omega$ and $k-\varepsilon$ zones without user interaction. The main additional complexity in the model formulation compared to standard models lies in the necessity to compute the distance from the wall, which is required in the blending function. This is achieved by the solution of a Poisson equation.

The SST model has greatly benefited from the strength of the underlying turbulence models, in particular the accurate and near wall formulation of the Wilcox model. The model was originally used for aeronautical applications, but has since made its way into
most industrial, commercial and many research codes. The proper transport equation for the SST model is thus given as:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \bar{F}_k - \beta \rho k \omega + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_{\omega} \mu_i) \frac{\partial k}{\partial x_i} \right] \\
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho U_j \omega)}{\partial x_j} = \alpha \rho S^2 - \beta \rho \omega^2 + \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_{\omega} \mu_i) \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_1) \rho \sigma_{\omega} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} 
\]

Where the blending function \( F_1 \) is defined by:

\[
F_1 = \tanh \left\{ \left[ \min \left( \frac{\sqrt{k}}{\beta \omega y}, \frac{500\nu}{\gamma^2 \omega}, \frac{4\rho \sigma_{\omega} k}{C_{D_{k/\omega}}} \right) \right]^4 \right\} 
\]

With \( C_{D_{k/\omega}} = \max \left( 2\rho \sigma_{\omega}^2 \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \) and \( y \) is the distance to the nearest wall.

\( F_1 \) is equal to zero away from the surface of the \((k-\varepsilon)\) model and switches over to one inside the boundary layer \((k-\omega)\) model. The turbulent eddy viscosity is defined as follows:

\[
\nu_t = \frac{a_k}{\max (a, \omega, SF_2)}
\]

Where \( S \) is the invariant measure of the strain rate.

\( F_2 \) is a second blending function defined by:

\[
F_2 = \tanh \left[ \max \left( \frac{2\sqrt{k}}{\beta \omega y}, \frac{500\nu}{\gamma^2 \omega} \right)^2 \right]
\]

A production limiter is also used in the SST model to prevent the build up of turbulence in stagnation regions:
Turbulence Modeling and Solution Strategies
Chapter 3

\[ P_k = \mu_k \frac{\partial U_j}{\partial x_j} \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right) \rightarrow \tilde{P}_k = \min \left( P_k, 10 \cdot \beta \rho K \omega \right) \]

All constants are computed by a blend from the corresponding constants of the \( k-\varepsilon \) model and the \( k-\omega \) model via \( \alpha = \alpha_1 F + \alpha_2 (1 - F) \). The constants for this model are:

\[ \begin{align*}
\beta & = 0.09 \\
\alpha_1 & = \frac{5}{9} \\
\beta_1 & = \frac{3}{40} \\
\sigma_{s1} & = 0.85 \\
\sigma_{e1} & = 0.5 \\
\alpha_2 & = 0.44 \\
\beta_2 & = 0.0828 \\
\sigma_{s2} & = 1 \\
\sigma_{e2} & = 0.856
\end{align*} \]

One of the essential features of a useful turbulence model is an accurate and robust near wall treatment. In addition, the solution should be largely insensitive to the near wall grid resolution. The requirement \( y^+ < 2 \) is excessive and may not be satisfied for all walls. On the other hand, the strict use of wall functions, which allow the use of coarser grids, limits the model accuracy on fine grids. Esch and Menter (2003) proposed a near wall treatment which automatically shifts from the standard low Reynolds number formulation to wall functions, based on the grid spacing of the near-wall cell. This new wall treatment formulation is part of SST model. Figure 3.5a shows velocity profiles for Couette flow simulations by Menter et al (2003) on three vastly different grids \( (y^+ \sim 0.2; y^+ \sim 9; y^+ \sim 100) \). Menter (2003) reports that despite the large differences
in the near wall spacing, the computed shear-stress varies by less than 2% and all solutions follow the logarithmic profile.

![Velocity Profile for Three different grids using SST wall treatment-Menter (2003)](image)

At the 2nd AIAA drag prediction workshop sponsored by NASA (http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw) a team of engineers from CFX (http://www.ansys.com/assets/testimonials) used the SST model to simulate aerodynamic flows past two wide bodied jets. Each team were giving two grid types; one with Re of 5.83 million (WB) and 8.43 million (WNBP) respectively, both made of hexahedral cells. Figure 3.5b and 3.5c shows the drag polar for the mandatory runs against the experimental data. The simulated results are in very good agreement with the
experimental data. This is a strong indication that an optimized RANS model like the SST is capable of predicting accurately aerodynamic properties.

Figure 3.5b Geometries for AIAA Drag Prediction Workshop WB (left), WBNP (right)

Figure 3.5c Drag Polar for AIAA Drag prediction workshop.
3.7 The Spalart-Allmaras (S-A) Model

The Spalart-Almaras model (1992) is a relatively new and simple one-equation model that solves the modeled transport equation for the kinematic eddy viscosity. It was designed specifically for aerospace applications that involves wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. The model solves a transport equation for a quantity $\tilde{\nu}$, that is identical to the turbulent kinematic viscosity except in the near wall or viscous affected region. The transport equation for $\tilde{\nu}$ is given as:

$$\frac{\partial}{\partial t}(\rho \tilde{\nu}) + \frac{\partial}{\partial x_j}(\rho \tilde{\nu} u_j) = G_\nu + \frac{1}{\sigma_y} \left[ \frac{\partial}{\partial x_j} \left\{ (\mu + \rho \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right\} + C_{\kappa 2} \rho \left( \frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right] - Y_\nu + S_\nu \tag{3.28}$$

The term $G_\nu$ is the production of the turbulent viscosity and is modeled as:

$$G_\nu = C_{\mu 1} \rho \tilde{S} \tilde{\nu}$$

where

$$\tilde{S} = S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{x2}$$

and

$$f_{x2} = 1 - \frac{X}{1 + X f_{x1}}$$

$C_{\mu 1}$ and $\kappa$ are constants, $d$ is the distance from the wall, and $S$ is a scalar measure of the deformation tensor and is based on the magnitude of the vorticity:
\[ S = \sqrt{2\Omega_y\Omega_y} \]

Where \( \Omega_y \) is the mean rate-of-rotation tensor and is defined by

\[
\Omega_y = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)
\]

The Spalart-Allmaras model is a relatively new turbulence model; it does not have a history of usage and application as the \( k-\varepsilon \) model. It is often criticized for its inability to rapidly accommodate changes in length scale, such as might be necessary when the flow changes abruptly from a wall bounded to a free shear flow.
3.8 Detached Eddy Simulation (DES) Model.

Experience has shown that the LES in boundary layer flows at high Re numbers is expensive. The DES is an attempt to combine elements of RANS and LES formulations in order to arrive at a hybrid formulation, where RANS is used in attached and mildly separated boundary layers and LES is applied in massively separated regions. While this approach offers many advantages, it is with problems, as the model must identify the different regions automatically and could be sensitive to grid generation or meshing.

The base model employed in majority of DES applications is the Spalart-Allmaras one-equation model by Spalart et al (1997) and an analogous model proposed by Strelets (2001) which is based on the SST RANS model described earlier. The idea behind the DES model of Strelets is to switch from the SST-RANS model to an LES model in regions where the turbulent length, $L_t$, predicted by the RANS model is larger than the local grid spacing, essentially the length scale used in the computation of the dissipation rate in the equation for turbulent kinetic energy is replaced by the local grid spacing $\Delta$.

The DES modification in the SST is applied to the dissipation term in the $k$-equation is given as:

$$\rho\varepsilon = \beta'\rho k\omega \rightarrow \beta'\rho k\omega \cdot F_{\text{DES}} \quad \text{with} \quad F_{\text{DES}} = \max \left( \frac{L}{D_{\text{DES}}\Delta}, 1 \right)$$

Where $\varepsilon$ is the dissipation rate, $\Delta$ is the maximum local grid spacing

$$(\Delta = \max (\Delta x, \Delta y, \Delta z))$$

in case of a Cartesian grid. $\beta'$ is a constant of the SST model,

$$L_t = \frac{\sqrt{k}}{\beta' \omega}$$

is the turbulent length scale and $C_{\text{DES}} = 0.61$ is a calibration constant of the DES. The practical reason for choosing the maximum edge length in the DES
formulation is that the model should return the RANS formulation in attached boundary layers. The maximum edge length is somewhat a safe estimate. Still one practical problem with the DES formulation is that there is no mechanism for preventing the limiter from becoming active in the attached portion of the boundary layer. Also the use of the grid spacing in the limiter may induce separation falsely.

3.9 Advection Schemes

In Computational Fluid Dynamics one of two numerical methods are normally used to solve the discretized Reynolds averaged Navier-Stokes equation of the domain:

- Segregated Solver.
- Coupled Solver.

The two numerical solution strategies use the same discretization process, their difference lies in the linearization approach and in how the equations are solved. Segregated solvers employ a solution strategy where the momentum equations are first solved, using a guessed pressure, and an equation for a pressure correction is obtained, as a result of this ‘guess and correct’ nature of the linear system, a large number of iterations are typically required in addition to the need for judiciously selecting relaxation parameters for the variables.

The coupled solver solves the governing equations of continuity, momentum, and energy simultaneously. This solution approach uses a fully implicit discretization of the equations at any given time step. For a steady state problem, the time-step behaves like an acceleration parameter, to guide the approximate solutions in a physically based
manner to a steady state, or to calculate the solution for each time step in a transient analysis.

In both the segregated and coupled solution methods, the non-linear governing equations are linearized to produce a system of equations for the dependent variables in all the computational cells. The mechanism by which the governing equations are linearized may be implicit or explicit. By implicit, it is meant that for a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore, each unknown will appear in more than one equation in the system, and those equations must then be solved simultaneously to obtain the unknown quantities.

For the explicit method, given a variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore, each unknown will appear in only one equation in the system and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

Solutions to the Navier-Stokes equations exist for the simplest of flows under ideal conditions. In order to obtain solutions for real flows, a numerical approach must be adopted whereby the equations are replaced by algebraic approximations which may be solved using a numerical method. This approach involves discretizing the spatial domain into finite volumes using a mesh. The governing equations are then integrated over each control volume. In figure 3.9, the geometric center which is denoted as $P$ is often referred to as the cell center.
In the CFX solver used here, all the dependent variables and material properties are stored at the cell center $P$. In essence the average value of any quantity within a control volume is given at its cell center. The generalized transport equation is often given as:

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \vec{V} \phi) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi$$

where $\Gamma$ = diffusion coefficient. $\nabla =$ gradient (of $\phi$).

This equation is known as the generic conservation equation for a quantity $\phi$.

Integrating this equation over a control volume cell, gives,

$$\int_\Omega \frac{\partial (\rho \phi)}{\partial t} d\Omega + \int_\Omega \nabla \cdot (\rho \vec{V} \phi) d\Omega = \int_\Omega \nabla \cdot (\Gamma \nabla \phi) d\Omega + \int_\Omega S_\phi d\Omega$$

For the purpose of this section, only the discretization of the convection term is considered which is given as follows:

$$\int_\Omega \nabla \cdot (\rho \vec{V} \phi) d\Omega = \sum \rho \phi \vec{V} \cdot \vec{n} A_e = \sum (\rho \phi \vec{V}_c) A_e$$
where subscript \( e \) denotes one of the faces of the cell in question, \( A_e \) is the area of face \( e \), \( V_e^\nu \) represents the velocity in the direction that is normal to the face. Because the solution variable is available at the center, various interpolation schemes with vary levels of numerical accuracy are in use. In the numerical simulations conducted, only the following were considered:

- 1\textsuperscript{st} Order Upwind Difference Scheme (UDS).
- Numerical Advection Scheme (Specify Blend).
- High Resolution Scheme.

For the UDS, upwinding means the cell face value is derived from upstream relative to the direction of the normal velocity \( V_e \). When first-order accuracy is desired, quantities at cell faces are determined by assuming that the cell-center values of any of the variables represent a cell-average value and hold throughout the entire cell.

In the specified blend, a variable value between 0 and 1 is used to reduce the diffusive properties of the UDS. When this value is equal to 1, it is considered to be 2\textsuperscript{nd} order accurate. The High Resolution Scheme computes the factor similarly to the specified blend locally, but ensures it's as close to 1 as possible while not violating the so called boundedness principle. The high resolution scheme is considered to be both accurate and bounded. The recipe for this procedure may be found in Barth and Jesperson (1989).
3.10 Parametric and Comparative Study on Turbulence Models.

It is a known fact that no one turbulence model possesses the universal characteristics of been superior for any class of problem. The choice of which turbulence model to use is dependent on many factors such as the physics inherent in the flow, established practice by prior researchers particularly in the industry and the available computational resources. To make the appropriate choice, knowledge of the limitations and capabilities of the each RANS model is essential. The purpose of this section is to review the results of simulations carried out on one of the bridge models used in this research with different turbulence models while comparing the results to the wind tunnel test results. The results of the exercise forms the basis of the choice of the turbulence model as well as the advection scheme used for the rest of this research.

For this exercise, consider the model of the Carquinez Bridge deck section under the West wind air flow at 0° angle of attack (See Chapter 6). Table 3.0 gives a matrix of the turbulence models, the advection schemes and the result of the wind tunnel tests for comparison. The parametric study consist of just one model but invokes the most commonly used turbulence models one at a time, so that the solutions are not mesh dependent. In the Shear Stress Transport (SST) model, the $k-\varepsilon$, and the RNG $k-\varepsilon$ model; three advection schemes are considered, namely, the upwind scheme, high resolution and specified blend factor. In a way, the high resolution scheme is an automatic form of the specified blend factor but with the solver left to determine the variation of what the value(s) of the blend factor to use throughout the domain based on the local solution field. All other values of the blend factor, is a blending between First
and Second order advection schemes to calculate the advection terms in the discrete finite volume equations, for example a value of 0.0 is equivalent to using the First Order Advection scheme, while a value of 1.0 uses a Second Order differencing for the advection scheme, therefore varying the values of the specified blend factor allows one to see the effect of the blending of first and second order schemes on the force calculation. The author did not run any simulation using the explicit $k-\omega$ based models because the SST model is an improved version and offshoot of the $k-\omega$ model and is judged to give superior performance.

For all intent and purpose, the SST model with the specified blend factor of about 0.8, which means a Second Order differencing scheme produced the result closest to the wind tunnel test.

In practice, it is often difficult to draw firm conclusions about turbulence model accuracy when performing multi-code CFD studies while using the same models because of inconsistencies in model formulation in different codes. The results of Table 3.0 are not multi code based. One of the possible contributing numerical errors in parametric study of turbulence models is how the model treats the near-wall effects. The near wall formulation determine the accuracy of the wall shear stress, in the $k-\omega$ based SST model as implemented in CFX, it automatically switches from a low Reynolds number formulation (here Reynolds number refers to the boundary layer) to a wall function treatment based on grid density, allowing for optimal performance of the turbulence model for any given grid. The $k-\varepsilon$ based models on the other hand includes near-wall damping functions to allow integration to the surface. Its accuracy is further improved
when the $y^+ < 1$. The other area of potential error is how the turbulence model handles numerical diffusion. Numerical diffusion depends on the order of discretization, gradient of the transport quantity and cell distance among others. The research does not delve into the subject of numerical diffusion, there is abundant information on this in the literature.

<table>
<thead>
<tr>
<th>Model: SST</th>
<th>ADVECTION SCHEME</th>
<th>$D$ (Drag,N)</th>
<th>$L$ (Lift,N)</th>
<th>$M$ (Moment,N-mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Upwind</td>
<td>0.019</td>
<td>-0.14</td>
<td>0.037</td>
</tr>
<tr>
<td></td>
<td>High Resolution</td>
<td>0.019</td>
<td>-0.046</td>
<td>0.012</td>
</tr>
<tr>
<td></td>
<td>Specified</td>
<td>0.95</td>
<td>-0.029</td>
<td>0.015</td>
</tr>
<tr>
<td></td>
<td>Blend</td>
<td>0.80</td>
<td>-0.052</td>
<td>0.014</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.75</td>
<td>-0.06</td>
<td>0.013</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.65</td>
<td>-0.075</td>
<td>0.012</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Model: $k-\varepsilon$</th>
<th>ADVECTION SCHEME</th>
<th>$D$ (Drag,N)</th>
<th>$L$ (Lift,N)</th>
<th>$M$ (Moment,N-mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upwind</td>
<td>0.022</td>
<td>-0.14</td>
<td>0.037</td>
<td></td>
</tr>
<tr>
<td>High Resolution</td>
<td>0.017</td>
<td>-0.127</td>
<td>0.032</td>
<td></td>
</tr>
<tr>
<td>Specified</td>
<td>0.95</td>
<td>-0.13</td>
<td>0.016</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.80</td>
<td>-0.12</td>
<td>0.016</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>-0.14</td>
<td>0.016</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.65</td>
<td>-0.15</td>
<td>0.016</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Model: (RNG) $k-\varepsilon$</th>
<th>ADVECTION SCHEME</th>
<th>$D$ (Drag,N)</th>
<th>$L$ (Lift,N)</th>
<th>$M$ (Moment,N-mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upwind</td>
<td>0.019</td>
<td>-0.148</td>
<td>0.039</td>
<td></td>
</tr>
<tr>
<td>High Resolution</td>
<td>0.014</td>
<td>-0.12</td>
<td>0.0294</td>
<td></td>
</tr>
<tr>
<td>Specified</td>
<td>0.95</td>
<td>-0.13</td>
<td>0.025</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.80</td>
<td>-0.10</td>
<td>0.020</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>-0.09</td>
<td>0.018</td>
<td></td>
</tr>
</tbody>
</table>

| Model: Spalart-Almaras      | N/A               | 0.026        | -0.14        | 0.026            |
| Wind Tunnel                 | N/A               | 0.027        | -0.047       | 0.015            |

Table 3.0 Comparison of Turbulent Models & Wind Tunnel tests at $0^\circ$ incidence.
3.11 Computational Costs of Turbulent Models.

Determining the computational costs of a turbulence model is dependent on many factors such as:

- Type of simulations whether laminar or turbulent
- High or Low Reynolds number.
- The size, type and volume of elements.
- Additional source terms included in the transport equations.
- Type of advection scheme employed in the solution strategy.
- If the solution algorithm is implicit or explicit.

In terms of computational costs, it is evident that the One-equation models such as Spalart-Allmaras will be the least expensive; this is because the number of variables in its one transport equation is all that must be resolved.

The standard $k-\varepsilon$ model definitely requires more computational time than the Spalart-Allmaras model because an additional equation must be resolved. The realizable $k-\varepsilon$ model and the RNG $k-\varepsilon$ model have extra terms and functions in the governing equations and a greater degree of non-linearity and hence took additional 20% CPU and time to solve based on the experience of the author in the simulation conducted.

The $k-\omega$ models are comparable in time and CPU usage to the $k-\varepsilon$ model. However, experience in this research suggests that the SST model during transient analysis takes an additional 7-10% more CPU than either the realizable or the RNG $k-\varepsilon$ models.

The Detached Eddy Simulation (DES) and the Large Eddy Simulations (LES) are basically in the same class of Turbulence model, while it is generally agreed that they are
far more accurate for medium to high Reynolds number simulations, they cannot be used for steady state solutions. Therefore, the formulations are only used for transient analysis that requires time step on the order of 1/1000 of a second. The LES is usually in the order of 2 or 3 time step lower than the DES model, making it very expensive and should only be considered in the rarest of situations or where there are ample computational facilities.

3.12 Concluding Remarks

Turbulent modeling and solution strategies that are most applicable have been presented. The ideal and preferred solution is the DNS, however its very expensive to implement, second is the DES-SST, while practical and reasonable to implement in comparison to the LES, it still requires a lot of sensitivity studies to ensure non interference of the grid spacing to stimulate unnecessary separation and or re attachment. Strelets (2001) proposed that a different numerical treatment should be employed in the RANS and in the detached flow regions of the domain when using the DES-SST, he proposed a switch between a second order upwind biased scheme for the RANS region and a second order central difference scheme for the DES region. The mechanism for determining this switch consistently is still unproven for industrial application on a consistent enough basis.

The large Eddy Simulation is equally attractive and elegant, but extremely expensive in terms of the time and computer resources that is required. For research studies involving 2-dimensional models, perhaps it may be worthwhile to use it; for the research undertaken here it is not practical. Galperin and Orszag (1993) in their review publication
suggest that typical applications today have been mainly for simple geometries. This is mainly because of the large computer resources that is required, of greater challenge is the lack of literature on the use of wall functions for the LES. Therefore, of the pure RANS model available, the SST model offers the most advantage. Its ability to switch between the robust $k-\varepsilon$ model and the substantially more accurate $k-\omega$ model in the near wall regions as well as been successful in capturing the true behavior of flows with moderate adverse pressure gradient, is of great advantage. It is also reasonably economical in comparison with the LES or DES models. Therefore, all simulations in this research will be based on the SST model.
4.0 Introduction

The process of grid or mesh generation is perhaps the most challenging aspect of applying Computational Fluid Dynamics to the solution for aerodynamic properties of a bluff body with details such as is encountered in a long span bridge deck section. The process is essential and vital to solving accurately the fluid mechanics equations either by the finite element, finite volume or finite difference method and is critical to obtaining good solutions.

To solve a represented problem numerically, the computational domain or control volume needs to be discretized into a collection of nodes or elemental volumes. These networks of discrete points or volumes are called grids and the process referred to as mesh or grid generation. The relevant and governing partial differential equations that serve as the basis of the Reynolds Averaged Navier Stokes (RANS) are solved numerically based upon these grids, to yield a solution over the domain. The most economically effective grid design will be influenced by a combination of the geometric configuration of the element(s) in the domain as well as the physical solution being done thereon. Hence, the grid generation becomes a critical issue, since it is at the grid points in the mesh where the critical desired parameters are being computed.
A turbulence model such as the Detached Eddy Simulations (DES) relies on the grid spacing as critical and important criteria in formulating a solution strategy. Therefore, the disposition of the grid points in the model to a necessary and sufficient extent dictates the validity and accuracy of the results in the simulations. And great deal of care and validation is required to get absolute quantities such as drag, lift and skin friction on bluff body such as a bridge deck section, hence grid generation is a necessary and critical step in the simulation process, and includes the bulk of the setup time for the problem.

The rate of stretching in the grid and the grid resolution in critical regions of high flow field gradients will affect the quality of the results. The number of grid points will dictate the CPU requirements and the computational time for the study. Since resources are usually limited (especially for this research), it is the intent of the grid generation to make the best use of the number of nodes that are available as well as the computing power available to do the simulations, and thus to make the grid points an active part of the numerical solution.

In this chapter, the types of grids and the important issues of grid generation are discussed. Additionally, the effects of grid types on the accuracy of the simulations are also studied.

4.1 Grid Element Types

In 3-Dimensional CFD analysis, there are essentially four widely used element types, namely:

- Hexahedral
- Tetrahedral
- Wedge or (Prism)
- Pyramid

Figure 4.1 shows the above element types and their associated degrees of freedom. Tetrahedral elements vary from 4 nodes to 16 nodes, with the 4-node elements most common. The basic hexahedral elements are 8-noded but can go up to 32 nodes. Wedges vary from 6 nodes to 24 nodes, with the 6-noded element the most used. Pyramids are essentially 5-noded elements and are mostly used to maintain conformability for transitioning from tetrahedral elements to hexahedral elements. It should be noted that the higher the number of nodes an element has the more expensive computationally it is, but higher order nodes do not necessarily correspond to better accuracy. In Computational Fluid Dynamics simulations, flows can generally be more easily be resolved by more elements of lower order than fewer elements of higher order.

![Figure 4.1a Tetrahedral Elements (4 and 16 nodes)](image)
Figure 4.1b Hexahedral Elements (8 and 32 nodes)

Figure 4.1c Wedge Elements (6 and 24 nodes)

Figure 4.1d Pyramid Element (5 nodes).

Reference Figures 4.1-4.d (Oden, Carey-1974)
4.2 Grid Types

There are essentially two types of grids, depending on the method of generation, the composition and type of elements used as described above, namely: structured and unstructured grids.

4.3 Unstructured Grids

Unstructured grids have their beginning in the Finite Elements (FE) world of structural modeling. The real introduction to Computational Fluid Dynamics came in the 1980s through the work of Weatherill (1988), Lohner (1988) and Baker (1989). Unstructured grids have inherent simplicity of construction in that, by definition, no structure is required. The process of constructing unstructured grids begins with a geometric definition of the domain to be meshed. Such a definition will be in the form of Non-Uniform Rational B-Splines (NURBS) curves and surfaces, or an equivalent, such as splines. Most unstructured grid methods then build a grid based on a hierarchical approach that involves generating grids on boundary curves, boundary surfaces and finally a volume grid. The shapes of the elements generated in unstructured grids can vary; traditionally, triangles on surfaces and tetrahedral in the volume have been used.
By their very nature, the irregular ordering of the connections between nodes within an unstructured mesh places great emphasis on techniques that enable searches to be
made through the grid in a fast and efficient manner. Hence, data structure plays an important role in the generation of unstructured meshes and in subsequent usage of such grids with solution algorithms. Techniques used in generating unstructured grids are, in most cases, based on relatively straightforward concepts. However, the practical application and implementation within a computer code is a major challenge.

With all the research activity devoted to automatic grid generation, there are now many techniques for the construction of unstructured grids. However, three approaches are widely used. They can be broadly described as:

- Tree Based Methods, such as Octree
- Point Insertion Methods based on Delaunay Triangulation.
- Advancing Front Methods.

4.3.1 Tree Based Methods

The tree based methods use a recursive subdivision method in the automatic generation of grids. The application of recursive subdivision over a spatial domain starts with a regular shape that is subdivided, in a regular manner, into a number of similarly shaped pieces, referred to as tree cells. The subdivision process is repeated until the smallest individual cells satisfy a given criteria.

The recursive subdivision provides a definite method of decomposing a geometric domain into a set of terminal cells that can be related to the grids or elements for use in the simulations. The associated tree structure also provide an effective means for
supporting various typical operations common to grid generation and numerical simulations, including determining the cells covering a particular location in space and determining cell neighbors. The process of determining the interactions of the cells of the tree with the analysis geometry and the decomposition of the cells into elements represents the most complex aspect of automatic grid generation using spatially based trees. However, in cases where the cells are directly allowed to represent whatever portion of the analysis geometry included within them, the grid generation process is very straightforward. In those cases where the elements defined in the tree cells have to conform to the geometry, the creation of elements in cells containing portions of the boundary of the domain is far more complex.

The application of recursive subdivision of a domain into subdomains, and the definition of an associated tree structure has a long history (Weimer & Warren 2002). There are a variety of means in which the domains can be subdivided and the associated trees defined. For the purposes of this discussion, emphasis will be on the Octree structure since the domain and models are three-dimensional. Octree structures have been used to support the development of three dimensional mesh generation for a number of years, see Baehmann et al (1987), Yerry and Shephard (1984). Although each of the Octree-based mesh generation methods is different, there are specific basic aspects common to all the procedures: the mesh generation process is implemented as a two step discretization process. In the first step the Octree is generated as shown in figure 4.3c, the tree is then used to localize many of the element generation process, in this process, the elements that will be generated in a part that falls within an Octree is governed by the criteria in that local region which constitutes the second step. Those cells (octants)
containing the parts of the objects boundary receive specific consideration to deal with the boundary of the object in relationship with the domain.

The input information required to generate the tree structure for use in mesh generation is the geometric model and information on the size of the elements desired in the domain. This aspect of the process requires careful consideration of the desired or expected results and to a larger extent the geometry or more precisely the minimum dimension on the object of study as shown below in figure 4.3c.

Figure 4.3c Typical Octant subdivision and determination of Model/Octant Interaction.
The information on desired element sizes will define the sizes of the terminal cells in the tree. This is accomplished by means of mesh control by prescribing a cell size as a function of the minimum dimension on the object. Since the edges of the terminal octants will become the edges of the elements in the grid, the size of the octants are dictated by the mesh control information applied. For a given root octant, the size of a terminal octant is controlled by its level in the Octree; therefore, the sizes of the elements are controlled by specifying octant root size and levels throughout the object being meshed. Since the Octree is, at least initially, spatially addressable, any mesh control function that can indicate the element size in a particular location in space can be used.

In general, three levels of mesh size specification are normally considered; the global mesh size specification, local mesh size specification and adaptive mesh size specification. The global mesh size specification uses global weight functions to control the mesh description over a domain. The local mesh size specification concept prescribes the desired mesh size variation on individual model entities. In the adaptive mesh size specification strategy, various mesh size sources (typically a background mesh) built according to the preceding problem analysis are used. Only the local mesh size control is directly related to the underlying model. The local mesh size specification consists of two parts; the required mesh size specification and the curvature-based mesh size control. The former concept is used to prescribe the mesh size explicitly at individual model entities, as shown in figures 4.4(a-d). The mesh size specifications are stored at each vertex (nodes) and at each control point of any curve or surface. These values are used to extract the mesh size specification at any location on a curve or surface. Moreover, each
model entity stores an upper bound limit on the mesh size which is not allowed to be exceeded.

Figure 4.4 Types and Effect of Mesh Control on Objects – from CFX.
The curvature based mesh size control is employed to enable accurate representation of a curve or surface by its discretization even if no particular mesh is required. The criterion is based on the ratio between the appropriate mesh size and the radius of curvature at a given location on the curve or surface.

The process of meshing the boundary cells is a strong function of the level of geometric complexity supported by the mesh generator. In cases where there is only a limited amount of geometric complexity allowed per octant, simple templates are possible. When there is no specific limitation on the level of geometric complexity allowed within the octant, the process of meshing the boundary octant requires all the functionality of an automatic mesh generator applied to the local region.

4.3.2 The Delaunay Method

The Delaunay method and the accompanying Voronoi diagram is one of the most popular methods of tetrahedral grid generation. The Delaunay-Voronoi methods only provide a mechanism for connecting nodes; it does not provide for the node creation, that is accomplished by a different method such as the Automatic Point Insertion and Local reconnection algorithm.

The Voronoi diagram sometimes known as the Dirichlet tessellation is the partitioning of a plane with $n$ points into convex polygons such that each polygon contains exactly one generating point and every point in a given polygon is closer to its generating point than to any other, this is illustrated in figure 4.6. The solid lines make up the Voronoi diagram forming a tessellation of the space surrounding the points. Each Voronoi tile, such as the
hatched area in figure 4.6b around point $P$ consists of the region of the plane that is closer to that point than any other.

![Figure 4.6 (a) Convex Polygon (b) Voronoi Diagram of a Planar set of Points.](image)

The Delaunay triangulation criterion, sometimes called the "empty sphere" property, simply stated, says that any node must not be contained within the circumsphere of any other tetrahedral within the mesh. A circumsphere can be defined as the sphere passing through all four vertices of a tetrahedron. Figure 4.7 is a simple two dimensional illustration of the criterion. Since the circumspheres of the triangles in (a) the other triangle's nodes, the empty circle property is maintained.
In figure 4.6b, the edges of the Voronoi diagram are formed from the perpendicular bisectors of the lines connecting neighboring points e.g., points $P$, $Q_3$, $Q_4$. This then determines a unique triangulation known as the Delaunay triangulation and is such that the circumcircle through each triangle contains no points other than its forming points.

In the automatic point insertion and connectivity optimization technique, point placement and connectivity is an independent process. For connectivity optimization, variations of the edge swapping or local reconnection algorithm of Lawson (1986) are used. In this scheme, the grid is repetitively connected to locally satisfy a desired criterion such as the Delaunay criterion or mesh control on any local member.

### 4.3.3 Advancing Frontal Techniques

The Advancing Front Techniques (AFT) are a family of related heuristic mesh generation methods for the finite element method suited for domains with complicated boundary curves and internal interfaces. The name refers to a strategy of generating
triangles sequentially from an ever shrinking set of dynamic curves that start at the boundaries and internal interfaces of the domain and advance to its interior. Lo (1985) used the method to produce a mesh triangulation in 2-D by linking a set of points, which had been generated before hand in a Cartesian fashion. Later, Peraire et al. (1987) used a new mechanism in which both the elements and the points were simultaneously generated. They also introduce grid control by allowing for specification of a spatial variation of the element size and shapes. This innovation was later extended to adaptive meshing in Computational Fluid Dynamics. Lohner and Parikh (1988), Peraire et al. (1988) subsequently extended the methodology to 3-Dimensions.

In the Advancing Front Technique, it is required that a surface mesher has already generated the surface mesh of the 3-D region or domain; the volume mesh is then generated from this front. The construction of a surface grid consists of approximating the surface by a set of planar triangular facets. This triangulation of the surface is a form of discretization of the surfaces of the object(s) and the domain into a general body conforming grid consisting of positioned points on the surface, which will constitute the nodes of the grid, and defining the links to be established between a node and its neighbors. The surface triangulation also entails subdividing the domain into consistent assembly of elements, this consistency means that the interaction between the elements occurs only on common points, sides and faces.

For the generation of the initial tetrahedral elements in the volume, the advancing front starts by assembling the initial triangular surfaces that are available from the initial domain boundary surface triangulation described earlier. A typical triangle generation
algorithm uses the concept of a generation front, the front is dynamic, in that the data structure is continuously changing during the generation process. When forming a new tetrahedron, the three nodes belonging to a triangular face from the front are connected either to an existing node or to a new node; any straight line segment that is available to form an element side is termed active, whereas any segment no longer active is removed from the front. During the generation process an active side is selected from the front and a triangular element is generated. This may involve creating a new node or just connecting to an existing one. After generating a tetrahedron, the front is updated. Thus while the domain boundary will remain unchanged, the generation front changes continuously and needs to be updated whenever a new element is formed. The generation process is completed when the number of triangles in the front is zero.

4.4 Structured Grids

Structured grids use curvilinear coordinates to produce a body fitted mesh. This has the advantage that boundaries can be exactly described and hence boundary conditions can be accurately modeled. In structured meshes, each interior nodal point is surrounded by exactly equal number of adjacent elements. Three directions can be identified within the mesh by associating a coordinate system called $\xi, \eta, \zeta$ system or IJK system with the mesh lines. The grid lines are the lines of constant $\xi$ or lines of constant $\eta$ or lines of constant $\zeta$. The node point $(\xi, \eta, \zeta)$ is formed with the intersection of grid lines $\xi, \eta$ and $\zeta$ as shown in figure 4.8a. Typically structured grids are composed of mostly hexahedral and or quadrilateral elements as shown in figure 4.8. Figure 4.8a shows the
numbering of nodes in a 2-dimensional structured grid layout, while figure 4.8b is the implementation in 3-dimensional.

Fig. 4.8a Structured Mesh Discretization and Organization of Nodes (2-D)

Fig. 4.8b Multi block Structured Grid (Courtesy ANSYS/ICEM-CFD)
There are two types of grid methods that apply to structured surface and volume grids:

- Algebraic Method

### 4.4.1 Algebraic Method

The algebraic grid generation techniques are a form of transformation of rectangular computational domain to an arbitrarily shaped physical domain. The Transfinite Interpolation (TFI) is one such method of carrying out the mapping, where the physical coordinates, treated as a function of the computational coordinates, are interpolated from their values on the boundaries of the computational domain. Since the data is given at a non-denumerable number of points, the interpolation is referred to as transfinite.

Consider the 2-dimensional (3-D will be similar) case of figure 4.9a, with a computational domain \([0,1] \times [0,1]\) with coordinates \(t\) and \(s\), and the physical domain with coordinates \(x\) and \(y\), to generate the grid in the physical space, a grid would be created in a unit square (2-D) or a unit volume (3-D) and this is then mapped into the physical domain.

In general there are two requirements for this to happen:

- The transformation must be on a 1-1 scale between the computational domain and the physical domain as shown in figure 4.9
- The boundaries of the computational space must map into the boundaries of the physical space
In the case of the 2-D, the interpolation is constructed as a linear combination of two 1-D interpolations or projections and their product. Soni (1985) suggests the use of blending functions that will be used to interpolate in each direction. If the blending functions \( \phi_d, \phi_t \) and \( \theta_d, \theta_t \) will be used to interpolate in each direction, then \( \phi_d \) interpolate from the \( s = 0 \) boundary into the domain, while \( \phi_t \) interpolates values from the \( s = 1 \) boundary into the domain. Similarly, \( \theta_d, \theta_t \) interpolates in the \( t \) direction. The requirements on \( \phi_d(s) \) and \( \phi_t(s) \) are:

\[
\begin{align*}
\phi_d(0) &= 1, \quad \phi_d(1) = 0 \\
\phi_t(0) &= 0, \quad \phi_t(1) = 1
\end{align*}
\]

and similarly \( \theta_d, \theta_t \) is constructed in the \( t \) direction.

In this case the simplest blending function will be linear giving the linear interpolation:

\[
\begin{align*}
\phi_d(s) &= 1 - s \\
\phi_t(s) &= s
\end{align*}
\]

For the \( x \)-coordinate, the 1-D projections \( (P_s, P_t) \) are formed as follows:

\[
\begin{align*}
P_s[x](s,t) &= \phi_d(s) x(0,t) + \phi_t(s) x(1,t) \\
P_t[x](s,t) &= \theta_d(t) x(s,0) + \theta_t(t) x(s,1)
\end{align*}
\]

and the product projection will then be

\[
P_s P_t[x](s,t) = \phi_d(s) \theta_d(t) x(0,0) + \phi_d(s) \theta_t(t) x(1,0) + \phi_t(s) \theta_d(t) x(0,1) + \phi_t(s) \theta_t(t) x(1,1)
\]

which is the finite interpolant for the values of \( x \) at the four corners of the computational domain. The 2-Dimensional transfinite interpolation is then:

\[
(P_s \otimes P_t)[x] = P_s[x] + P_t[x] - P_s P_t[x]
\]
which is then used to interpolate the entire boundary. To form the grid in the physical space, this interpolant is used to map the points of a regular grid in the computational space. The extension to 3-D models are far more intensive as the number of nodes and elements increases, most grid generation programs may take several hours or days to create a mesh, depending on the geometry.

Figure 4.9a Mapping of Boundaries and Transformation between Computational and Physical Domains (2-D).

Figure 4.9b Mapping of Boundaries and Transformation between Computational and Physical Domain (3-D)
4.4.2 Elliptic PDE Method

Elliptic grid generation is a technique of smoothing an initial (usually algebraic) mesh to improve grid quality (Thompson, Warsi, Mastin-1985). The grid improvements may involve forcing grid line orthogonality or forcing smooth grading of cell sizes. What makes elliptic grid generation challenging is that the grid smoothing must always ensure that the resulting grid points stay on the surface. Given this constraint, the effective approach of constructing a smooth grid is to work in the parametric space rather than on the physical surface.

This is accomplished by defining sets of second-order Partial Differential Equations (PDE) for the physical space coordinates in terms of the computational space coordinates, and then solving these equations on a grid in computational space to create a grid in physical space.

The elliptic system may preserve the original distribution of grid points or redistribute points based upon the choice of the control functions that are commonly used in adaptive grid generation. The control functions are evaluated either directly from the initial algebraic grid or by interpolation from the boundary point distribution and then smoothed. Orthogonality of the grid may be imposed along certain boundary components of the physical region. Boundary orthogonality can be achieved through Neumann boundary conditions, which allow boundary points to float along the boundary of the surface. Alternatively, the control functions can be determined to provide orthogonality at boundaries with specified normal spacing.
In its simplest form, consider the case of a 2-D grid generation. Here, a pair of Laplace equations is used to generate the mesh. As stated earlier in this methodology, rather than work in the physical domain where the geometry is complicated, it is preferable to work in the computational domain with simpler geometry but with far more complex equations. Once the work of mapping is done in the computational domain, the results are then projected to the physical domain. In a 2-Dimensional case (Figure 4.10), when transformed from the computational space \((\eta, \xi)\) to the physical \((x, y)\) space, the Laplace equations \(\nabla^2\eta = 0\) and \(\nabla^2\xi = 0\) which is the simplest solution normally used become:

\[
\begin{align*}
\alpha x_{\eta\eta} - 2\beta x_{\eta\xi} + \gamma x_{\eta\eta} &= 0 \\
\alpha y_{\eta\eta} - 2\beta y_{\eta\xi} + \gamma y_{\eta\eta} &= 0
\end{align*}
\]

where:

\[
\begin{align*}
\alpha &= x_\eta^2 + y_\eta^2 \\
\beta &= x_\xi x_\eta + y_\xi y_\eta \\
\gamma &= x_\xi^2 + y_\xi^2
\end{align*}
\]

The boundaries in the physical region will then be mapped to the computational plane to become boundary values for the two elliptic equations. The above equations will then be finite differenced in the computational space and solved iteratively to fill in the interior values. The results are then plotted as grids in the physical domain. The extensions to 3-D
are similar, with the solution best done via computer programs as it involves intensive
iterations to generate the grids for the physical domain.

![Grid Mapping Diagram]

**Fig 4.10 Mapping from Computational space to Domain Space.**

### 4.5 Grid Considerations for Turbulence Flow Simulations

Determining the grid spacing normal to a wall for a viscous analysis, particularly in
the zero velocity condition at the wall and at the resulting boundary layer velocity profile
that require fine resolution normal to the wall for proper modeling, involves several
factors. This normal spacing is a function of the flow condition at which the analysis will
be run and also a function of the length scale of the geometry in question.

For example for a wing, the reference length is usually taking as the root chord length. It
is also a factor of the flow solver parameters, such as turbulence model, and the
sensitivity of the algorithm to wall spacing. As noted in chapter 3, some turbulence
models such as the $k-\omega$ do accept coarser meshes since they have options such as automatic wall treatment (Section 6.5, Chapter 6) to correct for wall scale lengths as in this case.

![Diagram of boundary layer subdivisions](image)

**Fig 4.10b Subdivisions of the Near-Wall Region of a Boundary Layer-Full Velocity Profile**

The desired goal, whether through meshing or by correction in the turbulence model is to get enough resolution in the boundary layer to adequately define the boundary layer profile, and get reasonably accurate turbulence effects (depending on the study goals), without slowing the convergence excessively due to tight grid spacing. One method of assessing this spacing is through the calculation of the quantity called \( y^+ \). In turbulent flow, the boundary layer is defined as that thin region on the surface of a body in which
the viscous effects are important. The boundary layer (Fig 4.10c) allows the fluid to transition from the free stream velocity $U_t$ to a velocity of zero at the wall. In this layer, the velocity component normal to the surface is much smaller than the velocity parallel to the surface and the gradients of the flow across the layer are much greater than the gradients in the flow direction. This boundary layer thickness $\delta$ is defined as the distance away from the surface where the velocity reaches 99% of the free stream velocity.

$\delta = y$, where $u/U = 0.99$

Figure 4.10b shows the boundary layer structure, here the following variables used are defined thus:

$y$ = Distance from the wall.

$\nu$ = Kinematic Viscosity.

$U$ = Velocity at $y$. 
Friction Velocity, $U_f = \left( \frac{\tau_w}{\rho} \right)^{0.5}$

$\tau_w =$ Wall Shear Stress

The presence of walls as boundary conditions significantly affects turbulent flows. The velocity field is affected by the no-slip boundary condition at walls surrounding the bridge deck section (see Chapter 6). The near wall modeling and grid resolution in the vicinity of the boundary layer has impact on the solution and response of the model, since the walls are the main source of mean vorticity and turbulence. Therefore, accurate modeling of the flow in the near wall region will determine the prediction of wallbounded turbulent flows. The near-wall region can be divided into three-layers: the innermost layer, called the "viscous sub-layer" where the flow is almost laminar. Then there is the outer layer, called the fully turbulent layer where turbulence plays a major role. Lastly, is the interim region between both the viscous sub-layer and the turbulent region where the effect of viscosity and turbulence plays significant role. Generally, there are two methods for modeling near-wall regions. In one methodology, the viscosity dominated inner regions consisting of the viscous sublayer and the buffer layer is not resolved, rather semi empirical formulas called wall functions are used to transition between the viscosity dominated region between the wall and the fully turbulent region, hence there is no need to modify the turbulence models to account for the presence of the boundary walls. In the other method, the turbulence models are modified to allow the viscosity dominated region to be resolved with a mesh all the way to the boundary wall. As the name suggests, the wall functions procedure are just a collection of empirical
formulas that are used to link the solution variables at the near wall cells and the corresponding quantities on the wall. In the near wall modeling by meshing, the boundary layer is resolved by using very small mesh length scales in the direction normal to the wall, the variable \( y^+ \) is then the first grid spacing increment normal to the wall, measured in the units of the Law of the Wall and is based on flat plate boundary layers. Therefore, the modeling of turbulent flows requires some consideration during the mesh generation. For practical applications, the reference length is used in the \( y^+ \) calculation and a fixed spacing is usually applied to the wall, even though the thickness of the boundary layer changes as flow develops.

Equation 4.1 suggested by Bush (1988) from the flow solver NASTD and used in Aerospace Engineering is one appropriate estimation:

\[
\Delta y_{\text{physical}} = \frac{L}{R_e \nu} \left( \frac{V_{\text{wall}}}{\nu_{\infty}} \right) \sqrt{\frac{2 \rho_{\text{wall}}}{C_f \rho_{\infty}}} \tag{4.1}
\]

where \( L \) is the length scale used in \( R_e \) (\( L \) could be chord, diameter, body length or any other dimension); \( R_e \) is the Reynolds number, \( \rho \) is the density, \( \nu \) is the kinematic viscosity, \( \Delta y \) is the first cell height and \( C_f \) is the skin friction coefficient.

The subscript “wall” denotes values at the wall, and the subscript “\( \infty \)” denotes free stream values. If a better estimate is not available, a suitable value of \( C_f \) is 0.002.

The flat plate relationship is also useful:

\[
C_f = 0.025 \left( \frac{R_e \nu}{L} \right)^{1/3} \tag{4.2}
\]
Except for Hypersonic applications, it generally may be assumed that the ratio:

$$\left( \frac{v_{\text{wall}}}{v_m} \right) \text{ and } \left( \frac{\rho_{\text{wall}}}{\rho_m} \right) \text{ may be taken as } 1.0.$$

A rule of thumb in a course taken by this author on ‘Advanced Meshing’ (CFX) gives a relationship between the first cell height and \( y^+ \) as:

$$\Delta y = \sqrt{80 \cdot y^+ \cdot \frac{\mu}{U}}$$

(4.3)

where \( \Delta y \) = First cell height, \( y^+ \) = required \( y \)-plus, \( \mu \) = Dynamic viscosity and \( U \) = Free Stream velocity.

Using flat plate boundary layer theory, Schlichting (1966) also suggested this equation

$$y^+ = 0.172 \left( \frac{y}{L} \right) \text{Re}^{0.9}$$

(4.4)

Where \( L \) = body length and \( \text{Re} \) = Reynolds number based on the body length.

In determining the drag coefficient, it is normally assumed that the wall surface is smooth. In practice, it is rarely so. Grid spacing should be configured to capture the “roughness” on the bridge deck surface. In the current research, the wind tunnel report does not provide the roughness characteristics of the test model. A proposed strategy can be established however to enhance the determination of the roughness effects on the turbulent boundary layer.
Consider figure 4.11a, showing the characteristic roughness height which is derived as a function of the equivalent sand roughness within the viscous sublayer. The grid size at this level has to have a size in the order of 3-5 magnitude less than the roughness height in order to capture these phenomena.

In most situations as shown in Figure 4.11b, it is inevitable that the boundary layer may get separated from the solid body. This boundary layer separation will result in a large increase in the drag force on the body. Again the grids should be sufficiently fine enough to capture this behavior.

Fig. 4.11a Characteristic roughness height within the Viscous sub layer.

Fig. 4.11b Boundary Layer Separation.
To determine the significance and influence of the characteristic roughness height (a function of the equivalent sand roughness), consider the ratio of $\frac{k}{\delta_v}$, then two cases can be distinguished.

When $\frac{k}{\delta_v} \ll 1$, then $k$ does not affect the turbulent boundary layer significantly, so the 'smooth' assumption on the wall element would be valid. Conversely, when $\frac{k}{\delta_v} \gg 1$, then the 'rough' assumption will be essential.

A common initial guess for the boundary layer thickness for use in meshing the object of study and the domain is the relationship:

$$\delta_v = \frac{1}{\sqrt{R_e}}$$ Where $R_e$ is the Reynolds number of the flow.

Thus, for $R_e$ of $10^6$, the boundary layer thickness is of the order of $10^{-3}$. A far more elegant solution for the viscous sublayer thickness is obtained from that proposal by Launder and Spalding (1974), which yields:

$$\delta_v = \frac{\mu y_v^*}{\nu C_{mu} k_p^{1/2}}$$ where $y_v^* = 11.225$, $\mu$ = viscosity of the fluid, $\rho$ = fluid density, $k_p$ = turbulence kinetic energy and $C_{mu}$ = constant, approximately 0.09.

It is often necessary to refine the grid in the immediate vicinity of the leading and trailing edges of bluff bodies in external flows and such resolution will require sensitivity studies as some turbulent models may trigger false separation and improbable vortex shedding.
In near wall regions, boundary layer effects give rise to velocity gradients which are greatest normal to the surface. Computationally efficient meshes in these regions require that the elements have high aspect ratios, if tetrahedra are used, then a prohibitively fine surface mesh may be required to avoid generating highly distorted tetrahedral elements at the surface. It may be necessary to 'inflate' the local elements normal to the 2-D surface triangular elements into 3-D 'prism' elements at selected walls as shown below.

![Inflated mesh at Near Wall region(s)- from CFX](image)

**Fig. 4.12 Inflated mesh at Near Wall region(s)- from CFX**

### 4.6 Issues in the Grid Generation Process

This section looks at issues that arise in the grid generation environment for CFD applications. Creating a grid for a specific application is highly dependent on several factors. These include the detailed geometric features of the configuration to be analyzed, the grid generation and flow solver codes to be used, and the type of grids being generated; i.e. structured or unstructured grids. Grid generation is also subject to
management issues such as schedule, budget and resource allocation.

Setting aside the issue of schedule, the issues affecting grid generation boil down to disk space to store the grids and eventually the related flow solutions, and memory to compute the solution or smooth the grid.

Another issue that often arises is how much detail should be included for a particular study. For example, can complex geometry of a parapet on a bridge deck be approximated with an equivalent shape without losing the essence of the model. Issues of the quality of the mesh can broadly be categorized into the following:

- **Minimum Angle:** This criterion measures the minimum internal angle of the mesh as shown in figure 4.13a.
- **Maximum Angle:** This criterion relates to the maximum internal angle of the mesh as shown in figure 4.13b.
- **Minimum Edge:** This criterion relates as a ratio to the minimum edge length of elements whose shape functions are known to not cause numerical problems.
- **Maximum Edge:** This is a measure of the largest edge of each element. This criterion is checked by computing shape parameters often in the form of a Jacobian ratio, and then comparing it to default values whose shape functions and side ratios are known to pose no numerical instability.
Fig. 4.13 Undesirable Skew Elements in Grid Cells.

- Shape Quality: This quality criterion measures the likeness of the element to the reference or ideal one, regular tetrahedral in case of tetrahedrons and quadrilaterals in case of hexahedrons. Its value is 1 for a perfect element and it decreases as the element quality gets worse. If it becomes negative it means that the element have a negative Jacobian at some point which means that the curvature of the geometry is high and the element size is not small enough.

for a tetrahedra, the shape quality is defined as:

$$q = \frac{6\sqrt{2}}{5} \frac{\text{Volume}}{\sum_{i} l_i^3}$$  \hspace{1cm} (4.5)$$

where Volume = Volume of the tetrahedra and $l_i$ = the tetrahedra edges.
for hexahedra:

\[ q_n = \frac{\text{Volume}_n}{\left( \frac{l_1^2 + l_2^2 + l_3^2}{3} - \left( A_1^2 + A_2^2 + A_3^2 \right) \left( \text{Volume}_n \right)^\frac{1}{3} \right)^\frac{1}{2}} \]  

(4.6)

where \( l_1, l_2, l_3, A_1, A_2, A_3 \) are the lengths and areas of concurrent edges and faces of the element, while \( \text{Volume}_n \) is the volume of a fictitious parallelepiped made with the concurrent faces.

Another important criterion is body conformance of the grid. That is, one set of grid lines should always coincide with the physical boundary of the spatial domain regardless of the geometric complexity as reported by Choi (1997). He stated that this rule is often met, although it is very difficult to generate a boundary conforming grid for a highly curved surface. The results vary enormously even when there is a slight variation in the proximity of the grid line with the wall of the bridge deck section in this problem. As the fluid flows, the nose of the bridge section separates the flow and shears it sharply, thereby letting the wind pass by over its boundaries. Since the wind-flow process is very rapid and dynamic in nature, especially in the close vicinity of the boundary, the manner of distribution of grid points, on and closer to the boundaries of the bridge section makes a significant difference on the resulting pattern of vortices and coefficients. Therefore, a highly dense and coherent grid distribution is warranted over those important regions in order to capture the dynamic variation of lift and suction forces better. These effects are discussed in chapter seven.
Grid lines that intersect a boundary should intersect that boundary perpendicularly so that derivative boundary conditions can be easily implemented accurately. This is the orthogonality criterion. In some situations the velocity or pressure gradient normal to the wall surface may be equal to zero, these boundary condition can be easily implemented with very low error if the grid lines intersect a boundary orthogonally. In the interior of the spatial domain, the angle of intersection between grid lines only needs to be nearly orthogonal; somewhere between 45 and 135 degrees is acceptable.

The aspect ratio of the computational cells is an additional issue that arises during the setup of the computational grid. While large aspect ratios may be acceptable in some problems, a general rule of thumb might be to avoid aspect ratios in excess of 5:1. This limit can be acceptably exceeded when the gradients in one direction are very small relative to those in a second direction. Excessive cell aspect ratios can lead to stability problems, convergence difficulties, propagation of numerical errors and significantly increase the computational effort. Selvam (1994) evaluated the performance of various solution procedures in terms of CPU usage and the number of iterations for various aspect ratios of 1, 10, 60 and 160. Selvam reports that as the aspect ratio increases there is significant rise in the CPU time and number of iterations.

A dominant source of errors in multi-dimensional situations is the so-called false diffusion or numerical diffusion. Numerical diffusion is usually exhibited by difference equations where the advection term has been approximated by using an odd order scheme. A lot of CFD simulations employ an Upwind Differencing Scheme (UDS) which
is known to introduce this false diffusion error, upwind(ing) means that the face value of
the quantity being convected is derived upstream relative to its normal velocity. This
Upwind Difference Scheme is either of the First Order or the Second Order type for
solving the advection terms. When the First Order Upwind Scheme is used, the quantities
at the cell faces are calculated on the assumption that the cell-center value of any of the
field variables is represented by the cell average value and holds throughout the entire
cell. If the flow is aligned with the grid, a First Order Upwind Scheme is usually
acceptable for a quadrilateral or hexahedral grid however when the fluid flows at
variance with the grid, then the First Order scheme will introduce numerical diffusion
errors, except perhaps if the grid is extremely fine which in turn becomes expensive in
terms of computational resources. In Second Order Upwind Scheme, the assumption of
piecewise constant cell distribution is replaced by a linear distribution, this allows for
higher-order accuracy at the cell faces to be obtained using a Taylor series expansion of
the cell centered solution about its centroid. The Second Order Upwind scheme is known
to provide lower numerical diffusion error and is much more robust since it is based on a
much more accurate finite difference stencil. Beyond the strict first or second order
option of the Upwind scheme is the Specified Blend factor, the Blend factor, $\beta$ is actually
a Numerical Advection Correction factor meant to reduce the diffusive properties of
Upwind Differencing Scheme and it varies between 0 and 1. If $\beta = 1$ then the Second
Order Differencing scheme is invoked and if $\beta = 0$ the First Order Difference is invoked,
the choice of what values to use is very difficult to make, however the CFX-Solver used
for this work provides a switch option termed ‘High Resolution’ which then calculates
the $\beta$ value based on the work of Barth & Jesperson (1989). The proper calculation of $\beta$
using this procedure is automatically done and varied in the domain, for example it might
be zero near discontinuities and in the free stream where the solution has little variation.
This strategy therefore reduces the errors coming from numerical diffusion in the
simulations.

Proper grid alignment requires that one set of grid lines should align with the flow
direction according to Choi (1997). This condition is important for convection dominated
flows when the aspect ratio of the control volume about each grid point is very high and
or when the Navier-Stokes equations are used to study such flows. Here, the grid lines are
aligned in the direction of the wind flow over the bridge.

To minimize an extremely large number of grid cells and at the same time maintain a
sufficient degree of accuracy in the solution, a non-uniform grid is used. In a non-uniform
grid, the grid spacing is minimized in regions where high gradients are expected and
increased in regions where the flow is relatively uniform. The spacing between grid
points should change slowly from a region where grid points are concentrated to a region
where grid points are sparsely distributed. That is, the rate of change of grid spacing
should be minimized. Normally, the spacing between adjacent grid lines should not
change by more than 20% or 30% from one grid line to the next (Ansys-CFX Theory,
2003). This is an accuracy consideration, primarily impacting the accuracy of the
diffusion terms in the governing fluid flow transport equations. This condition is
important because otherwise, the numerical solution procedures in the computations may not be stable and robust and the solutions may start to diverge uncontrollably. Choi (1997) also reports that Fourier components which make up the solution reflect and refract at interfaces where grid spacing changes disproportionately.

The total number of grid points in the grid should be kept to level needed to obtain solutions of the desired accuracy. This condition should be met for either the structured and unstructured mesh and is very significant for computational efficiency. This can be achieved by clustering grid points in the region of interest where they are most needed and reducing the concentration elsewhere. Where large gradients are expected, as in shear layers, the grid should be fine enough to minimize the change in the flow variables from node to node. If the grid points are not clustered in the regions where needed, the solutions obtained may not have meaningful physics due to a low accuracy. The classical example is in the boundary layer computation, if not enough grid points are used where a boundary layer is expected to occur, the layer will not be visible. In this research, the limiting criterion is controlled to a great extent on the amount of computers and their memory availability.

The shape and size of the grid element is very critical in light of the computational accuracy needed by the solver code. The elements should have a reasonable aspect ratio. When the shape of the quadrilateral element is very skewed and slender, and also if the area of the grid cells are too small, then the Jacobian computations in the solution strategy becomes very difficult. The number of time steps has an inverse relationship
with the computational effort, the lesser the time step for discretization solution the more the number of runs to be computed per time bases and hence more computational effort and CPU time and vice versa. Skewed elements will require very small discretization time step increments for the transient solution to converge, thereby increasing the computational costs.

It is almost impossible to generate one single grid type that would satisfy all of the conditions listed above at every part of the spatial domain consistently, especially as the deck sections are rotated for a different angle of attack. Grid generation is an engineering skill and not academic, it is dependent on the experience of the engineer, and the software(s) available. The grids generated and used in this research are of good quality since they met all the afore mentioned criteria. Two grid generating software were used for this exercise: Gridgen (v.14.06) and CFX-Build (v.5.6) which is based on the Patran pre-processor.

4.7 Grid Generation Procedure

The process of grid generation is an engineering skill developed with experience, intuition, an understanding of bridge design and engineering and familiarity with both the grid generation software and the requirements of the solver that will be used for the simulation.

The grid generation process for the mesh(s) used in this research is best illustrated with the aid of the flow chart diagram in figure 4.14. The basis of the models are the
redrawing in AutoCAD (R2000) of the details of the super structure bridge deck section and scaling it to match the 1:250 models used for the wind tunnel test (Chapter 6).

Geometries created from details to be used in CFD simulations have to be 3-D solid elements made up of continuous curves and non interrupted surfaces. In this case, a 2-D section of the deck is first created using polylines, polylines are continuous lines composed of one or more line segments. The 2-D section is then extruded into 3-D and exported as ACIS.

The ACIS solid model files generated through AutoCAD are not suitable for meshing to the CFD standard for meshing or simulation. Quite often, there are gaps in the model, or the tolerance between connecting lines is too great, but most importantly the mesher still cannot make use of the file in the current format and it needs to be translated into IGES or parasolid. The IGES (Initial Graphics Exchange Specification) is an ANSI standard neutral spatial data format for exchanging CAD data between different software systems. This process is perhaps the most important in the geometry creation, without which the surface can not be meshed.

The following is accomplished in the process:

- Identifying and correcting serious surface definition problems.
- Creating missing geometries.
- Setting and meeting tolerance / deviations for the intended mesher.
- Stitching the models.

The IGES file so created is then imported into either CFX-Build or Gridgen to begin the process of creating the 3-D domain around the section. For the CFX-Build mesher,
the IGES file must still be converted to B-rep solids (Boundary representative) before the domain is built. The creation of the 3-D domain is the next most difficult to accomplish, primarily because both the deck section and the domain must share the same wall, this prevents fluid from flowing through the side of the deck, in essence there can be no gaps between the walls.

The process is largely dependent on the tools available in the grid generation software. In this research, the preferred method is to construct curves up to the boundary of the deck section; surfaces are then formed from the curves and mirrored in the opposite direction. Solids can subsequently formed from the two surfaces, as the angle of attack increases and the deck section has to be rotated from its geometric neutral axis, the process becomes more complicated especially for the curved or rounded edge detailed deck sections. These processes are termed ‘sewing’, since both the domain and the deck geometry are created independently and from different sources. Most grid generating software comes with a limited CAD application and Boolean operation options as compared with a full CAD program such as AutoCAD, this lack of options increases the difficulty of the geometry creation.

Following the domain creation, grid points are methodically located and distributed on the curves making up the deck section. In the case of hexahedral grids, this introduces another challenge, as the points on a curve must match any other curve(s) opposite it. For a tetrahedral mesh, this rule is not strictly enforced except when mesh control are introduced to emphasize areas of special considerations such as leading and trailing edges or around parapets.
For unstructured domains made largely of tetrahedrons, the criteria for either the Delaunay triangulation method or the Advancing Front technique can now be set. The criteria may include:

- **Surface shape parameters** – These attributes define how the shape of the surface grids is to be maintained.

- **Relaxation parameters** – These control the coefficient to be applied to a smoothing sweep through the grid points which is performed after all grid point insertion is completed. The smoothing algorithm works as follows:

  \[ r_{\text{smooth}} = (1 - \omega) r_{\text{original}} + \omega r_{\text{average}} \]

  where \( r_{\text{average}} \) = average value of all of the grid point's neighbors and \( \omega \) = relaxation parameter varying between 0 and 2.

- **Minimum / Maximum size of the triangle cell that will be allowed in the grid.** Normally this will be found by computing both the minimum and maximum arc length along the domain's connector and determining the area of triangle that has that spacing for each of its three sides.

- **Boundary Decay Factor** – This criterion affects how far into the grid interior the boundary cell size affects the interior cell size.

The grid generation may now commence while iteratively modifying the criteria's until the meshing is complete.

Grid generation for structured domains requires a generating path of either Algebraic methods using the transfinite interpolation (TFI) or the Elliptic PDE method. The TFI methods are based on the closed form algebraic equations that use the grid points on the
boundaries of the surface to calculate grid points on the interior. Several TFI methods are available such as:

- **Standard TFI** - Applies blending function based on the relative spacing of the grid points on the boundaries, see Soni (1985).

- **Linear TFI** - This applies the TFI algorithm to the (x, y, z) coordinates of the grid, but the blending functions are no longer related to the distribution of grid points on the boundaries. This method is rarely better than the standard TFI method.

- **Ortho TFI** - Uses a generalized form of the TFI equations allowing the specifications of first derivative values at the grid boundaries. First derivative values are computed on the boundaries so grid lines intersect the boundaries orthogonally with spacing interpolated from the adjacent boundaries.

The elliptic PDE method allows for the improvement of the smoothness of the structured domains while at the same time controlling orthogonality and clustering. The elliptic PDE method uses a similar relaxation parameter to the unstructured mesh as discussed above. The above procedures are followed iteratively with adjustments as necessary in the criteria until an acceptable mesh is created. Depending on the solver for the numerical simulation the boundary conditions may now be added.
Fig. 4.14 Grid Generation Process (As used in present work)
4.8 Comparison & Assessment of Grid Types

The principal advantage of the unstructured mesh generating approach is that it provides a powerful tool of discretizing domains of complex shapes and offers geometric flexibility since the number of neighbor nodes and elements are not predetermined. Structured grids however, are far more difficult to create primarily because they are body conforming. Most CFD solver codes find it convenient to solve structured meshes because they are optimized for the structured layout of the grid; they are also advantageous in data storage and memory usage. In this research, all the generated grids are structured. As shown earlier, unstructured grids can be tetrahedral dominated or hexahedral dominated. A tetrahedral dominated mesh hardly ever has hexahedral elements, while hexahedral grids commonly needs tetrahedrals and pyramids in transition areas where an hexahedral will not fit due to geometric constraints. The effect of element type used (Tetrahedral – fig. 4.15a or Hexahedral – fig. 4.15b) in turbulent external flow simulations has not received much attention from researchers. In fact, this author was unable to locate published research on the subject.

Given the orientation of the nodes, numbering of elements and the geometric properties of each element type, it is desirable and indeed necessary to determine which of the two is most accurate for these type of studies. Therefore, in this section a comprehensive study was undertaken to compare two different grids and their ability to predict the standard aerodynamic properties of the Bridge Deck i.e. Lift, Drag and Moment Coefficients when compared to the wind tunnel tests (see chapter 7)
In this exercise, consider a West Wind (see Chapter 6 for details) at an angle of attack of 2.5°. Several characteristics of the grid properties are compared in Table 4.1, while the ratio of total elements is almost 3:1 in favor of the Tetrahedral dominated mesh, the total number of nodes is virtually identical. As shown in Table 4.1, no hexahedral elements were used in the tetrahedral dominated mesh. For now, flow profiles around the deck section and the influence/effect of the grid type(s) are not discussed (see Chapter 7). It should be noted that for the models described, the wall resolution \((y^+)\) are fairly identical.

<table>
<thead>
<tr>
<th></th>
<th>Tetrahedral Dominated</th>
<th>Hexahedral Dominated</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>No. of Nodes</strong></td>
<td>3,626,634</td>
<td>3,650,657</td>
</tr>
<tr>
<td><strong>Number of Elements</strong></td>
<td>12,520,512</td>
<td>4,610,329</td>
</tr>
<tr>
<td><strong>Number of Tetrahedrons</strong></td>
<td>8,588,934</td>
<td>754,037</td>
</tr>
<tr>
<td><strong>Number of Prisms</strong></td>
<td>3,931,552</td>
<td>70,688</td>
</tr>
<tr>
<td><strong>Number of Pyramids</strong></td>
<td>26</td>
<td>1,034,382</td>
</tr>
<tr>
<td><strong>Number of Hexahedrons</strong></td>
<td>-</td>
<td>2,751,222</td>
</tr>
<tr>
<td><strong>Number of Faces</strong></td>
<td>791,628</td>
<td>727,694</td>
</tr>
</tbody>
</table>

*Table 4.1 Model properties & comparison of grid types.*

Next, a comparison of the number of prisms and pyramids; as expected prisms are dominant in the Tetrahedron model, an almost ratio of 56:1. As would be expected the number of faces is also virtually identical, so then the models provide for a good comparison and a means to determine a most accurate element type for this research.
study. The total number of elements is also of significance, the number of elements is such that they can not be generated on any available single CPU computer, hence the need for parallelization and partitioning of the models, further details may be found in chapter 5. Table 4.2 gives the details of the partitions for each model, following the MeTiS multilevel k-way algorithm discussed in chapter 5.

Fig. 4.15a Plane through Tetrahedron dominated model. Mesh concentration at leading and trailing edge, and around post show use of Mesh Control. Also note inflated surface at soffit and carriage way of deck section.

Fig. 4.15b Plane through Hexahedron dominated model. Mesh Control do not apply. Both trailing and leading edges are identical.
The simulation was carried out using the turbulent Shear Stress Transport (SST) model with the specified blend factor option for the advection scheme. In the CFX Solver, convergence is measured by the level of the RMS (root mean square) residuals; the amount by which the discretized equations are not satisfied and not the error in the solution, for both grid types this value are identical. The time of simulation varied between the two models; the Tetrahedron dominated models running for 8 days while the...
Hexahedron dominated models ran for 7 days. Table 4.3 shows the comparison between the two grid types as well as the wind tunnel test. Further comparisons may be found in Chapter 7.

<table>
<thead>
<tr>
<th>Variable (Coefficient)</th>
<th>Wind Tunnel Test</th>
<th>Tetrahedron Model</th>
<th>Hexahedron Model</th>
</tr>
</thead>
<tbody>
<tr>
<td><em>Drag, C_d</em></td>
<td>0.17</td>
<td>0.13</td>
<td>0.15</td>
</tr>
<tr>
<td><em>Lift, C_L</em></td>
<td>-0.075</td>
<td>-0.065</td>
<td>-0.0766</td>
</tr>
<tr>
<td><em>Moment, M_c</em></td>
<td>-0.0115</td>
<td>-0.0099</td>
<td>-0.0121</td>
</tr>
</tbody>
</table>

*Table 4.3 Comparison of Tetrahedron, Hexahedron and wind tunnel test.*

From table 4.3, it can be seen that the hexahedron dominated mesh produces a solution that is significantly closer to the wind tunnel test. It can be deduced therefore that given the same number of nodes, hexahedrons are advantageous over tetrahedrons. The primary reason is probably due to the fact that nodes for hexahedrons are better aligned with the flow, although solutions are carried out at the nodes, for external flow of this type, aligned nodes allows the solver to capture the near wall and shear layer forces better. The other reason, far more probable and significant, is in the gradient calculation which plays a significant and major role in CFD solutions. Gradient calculations are mainly done by the Green-Gauss approach in most solvers, the Green-Gauss theorem states the surface
integral of a scalar function is equal to the volume integral (over the volume bound by the surface) of the normal gradient of the scalar function, the algorithm used is dependent on cell shape and behaves better for hexahedrons than tetrahedrons. Nonetheless, grids generated and used for this research are now going to be hexahedron dominated.

4.9 Parametric Study on Hexahedral Grid Resolution.

As stated above, the grid of choice is Hexahedral, it stands to reason then that for computational economy and the accuracy of the simulations, additional parametric studies needs to be carried out. The study is needed to determine whether the number of mesh elements in the domain plays any significant role in the response of the models. For the parametric study, three models are studied, Model A being the original wind tunnel study model of the Carquinez Straits Bridge (Model 1 – Chapter 6) and two additional variations with modified mesh statistics as shown in Table 4.4 below. The study is about the spatial distribution of the cells in the domain and not about the modeling of the boundary wall/bridge deck interface which is expressed in the $y^+$ function as described earlier, that has been held constant.
Table 4.4 Mesh Statistics for Parametric Studies on Domain Grid Resolution.

<table>
<thead>
<tr>
<th>Mesh Statistics</th>
<th>Model A</th>
<th>Model B</th>
<th>Model C</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Nodes</td>
<td>3,844,950</td>
<td>2,486,129</td>
<td>1,762,994</td>
</tr>
<tr>
<td>No. of Elements</td>
<td>4,594,907</td>
<td>2,969,265</td>
<td>2,096,077</td>
</tr>
<tr>
<td>No. of Tetrahedrons</td>
<td>684,829</td>
<td>443,736</td>
<td>307,855</td>
</tr>
<tr>
<td>No. of Prisms</td>
<td>10,119</td>
<td>7,991</td>
<td>6,751</td>
</tr>
<tr>
<td>No. of Pyramids</td>
<td>867,408</td>
<td>561,833</td>
<td>392,457</td>
</tr>
<tr>
<td>No. of Hexahedrons</td>
<td>3,032,551</td>
<td>1,955,705</td>
<td>1,389,014</td>
</tr>
<tr>
<td>No. of Faces</td>
<td>748,795</td>
<td>491,820</td>
<td>349,946</td>
</tr>
</tbody>
</table>

Table 4.5 Comparison of Static Forces for Parametric Studies on Domain Grid Resolution.

<table>
<thead>
<tr>
<th>Variable (Coefficient)</th>
<th>Wind Tunnel Test</th>
<th>Model A</th>
<th>Model B</th>
<th>Model C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag, $C_d$</td>
<td>0.143</td>
<td>0.129</td>
<td>0.125</td>
<td>0.122</td>
</tr>
<tr>
<td>Lift, $C_l$</td>
<td>-0.188</td>
<td>-0.1857</td>
<td>-0.1944</td>
<td>-0.211</td>
</tr>
<tr>
<td>Moment, $M_C$</td>
<td>0.0174</td>
<td>0.0154</td>
<td>0.0187</td>
<td>0.0177</td>
</tr>
</tbody>
</table>
From Table 4.5, it can be deduced that the finer the grid resolution in the domain the more accurate the solution is. While the maximum variation is approximately 12% off from the wind tunnel test, there is also a time and resource saving on the simulation. While Model A ran for upward of 8 days on 4 computers, Model B took 5 days on 4 computers and Model 3 ran for 4 days on 3 computers. However, as will be shown in chapter 7, the Models B & C fail to resolve the flow in the domain particularly on the issue of vortex shedding. A phenomenon that is critical in the stability study of bridge deck sections. It can be concluded that for this work, grid resolution will not introduce a large amount of error and its not influential to the choice of the Turbulence Model.

4.10 Summary

This chapter reviewed the different element types available for grid generation, followed by a discussion of the generic criteria for generating grids to be used in external flow simulations such as in this research study. Grid types were also broadly classified and the methods of generating them summarized. The experience from this research suggests that turbulent modeling and grids are closely linked, to facilitate solutions that are realistic, suggestions and ideas were given to capture the aerodynamic qualities; e.g. forces that are often required in a fluid – structure simulations that involve external flows. Parametric study on the grid resolution suggests that while the accuracy improves with more grid points; roughly on a ratio of 2:1 in grid volume results in about 12%
off from the wind tunnel test result on the static forces, however this is not usually the only thing needed in the aerodynamic studies of bridge deck sections.

Finally, these ideas were tested in generating a set of two types of grids to establish the mechanisms and solutions strategy for the forces in this research. The numerical solution using the hexahedral grids proved superior and compared favorably with the wind tunnel experiments. In this parametric study, the solution time also favors the Hexahedron dominated grids by an average of slightly more than one day in eight. Therefore to large extent the comparison of the grid types, in this case hexahedrons and tetrahedrons, was a difficult diversion to make, for obvious reasons that tetrahedrons are much easier to create. However, the improved accuracy in the solution is worth the time and effort involved in creating and using hexahedrons.
5.0 Introduction.

In chapter 4, the principles, types and methods of grid (mesh) generation were discussed. Given the size or volume of grids needed to resolve the flow types and patterns associated with a typical steady state or transient simulation, as well as the need to obtain the aerodynamic forces from a typical computational fluid dynamics simulation, it is necessary to discuss the methodology and means of solving such large problems. Part of the attraction of numerical simulation is the economy of fast computers; however, as will be demonstrated here, the architecture and configuration of personal computers, though inexpensive, does not necessarily lend itself to solving large numbers of equations or models efficiently.

For example a 32 bit pc with 2 Gigabyte of RAM in the best case may be able to solve 1 million volume of hexahedral mesh. When the size of the mesh gets bigger for reasons enumerated earlier, it may be necessary to run the computers in parallel. This allows the discretized model to be broken into parts that can be solved within the limits of the architecture and memory capacity of the computers used. It is also necessary that the assemblage is maintained as if it is been solved on a single computer. The discussion here does not apply to main frame computers since they are normally configured for larger memory capacities. It should also be noted that the method and procedures discussed here are those used by the author for this research. Variations accounting for solver type,
computer type and architecture as well as operating systems should be duly noted before being applied elsewhere.

5.1 Computer Hardware & Architecture

Figure 5.1a below and Figure 5.1b on the following page shows the configuration and setup of the cluster built by the author for the simulation needed in this research. The master node is an Intel Pentium IV class computer with a 2.8 GHz Intel Processor and 2 Gigabyte of RAM. Slave nodes 1 & 2 are both Intel Pentium IV class computer but running at 2.6 GHz. The last slave node 3 is running at 2.53 GHz.

![Cluster Layout](image)

**Fig 5.1a Cluster Layout (Slave Nodes Connection to KVM omitted for clarity)**

The basic hardware needed in each computer node to build the cluster as shown in figure 5.1a:
• A network interface card (NIC) or Ethernet card.
• A networking switch or networking hub.
• Ethernet cable(s).
• A KVM (Keyboard, Video, and Mouse) switch board.

The Ethernet card provides a standardized way of connecting the computer nodes together to create the network. The networking switch or hub is akin to a traffic cop, directing and permitting 2 way traffic flow and exchange of data across the LAN between the master node and each of the slave nodes. The Ethernet cables are essentially coaxial cables capable of transmitting up to 10 Mb/s and are for connecting each computer node’s Ethernet card to the hub. The KVM switch is hardware device that enables the use of a single keyboard, video monitor and mouse to control all the computer nodes in the cluster.

Fig. 5.1b Cluster Nodes for Parallel Simulations.
5.2 Computer Nodes Communication

As indicated earlier, the network switch or hub permits data traffic between the computers in the cluster and for this to happen, there is a need to establish a protocol and a mechanism to accomplish this. The nodes consist of a master and the others as slaves and, as the name suggest, the master node is the controlling node from which the traffic and simulations are initiated, and for this work is the fastest computer; a 2.8 GHz Intel Pentium 4 class computer with 2 gigabyte of RAM. The other computers are slaves. Communication is established between the nodes by assigning a unique name and an IP address, the IP (Internet Protocol) defines how information or data gets sent between the master node and the slave nodes. The typical assigned IP address are as follows: 192.168.0.1 : 192.168.0.4, accounting for the 4 computers in the cluster.

5.3 Operating System Configuration

Linux is the operating system of choice for building PC clusters, primarily because of the low overhead in memory usage; in addition, there are kernels that allow the use of more than 2 Gigabyte of memory for a 32-bit machine, which is a real advantage over a comparable Windows based system. While the cluster built for this research allows the use of windows operating system for a Windows-Linux combination or an all Windows or all Linux system, the Linux based system offers the best economy, albeit at somewhat slower computational speed. Each computer node participating in the
clustering must have identical login names and passwords. Next either the *SSH* (Secure Socket Shell) or *RSH* (Remote Shell) daemon is configured on each node in the cluster, to permit communication between the master node and the slave nodes. A hidden file named `.rhosts` with the name of the master node or any other nodes with permission to remotely login is placed in each of the slave accounts. On a Windows machine, the IP addresses of the computers are also included, this normally being in the `\etc` directory of the operating system. Additional modifications for the Linux system includes turning on the remote login (rlogin) as well as the remote execute daemon (rexec).

### 5.5 Parallel Computing Software

In order to run in parallel, the computer hosts must be able to communicate with each other without passwords, and start and run the CFD solver using the assigned partition from the model that is to be solved. This process during the parallel run is accomplished by one of three ways:

- Parallel Virtual Machine (PVM)
- Message Passing Interface (MPI)
- Multi Computer Operating System for Unix (MOSIX)

The MPI by itself is simply a specification for message passing libraries, designed to be a standard for distributed memory, message passing and parallel computing. One such software system meeting this standard that is commonly used for accomplishing the MPI is MPICH which is freely distributed by Argonne National Laboratory through the office of Advanced Scientific Computing Research of the U.S Department of Energy; further
information on installation, compilation and the procedure for implementation on a Linux or Windows machine may be found at their website at: http://www-unix.mcs.anl.gov/mpi/mpich/.

In the PVM concept, the processes of a parallel run are distributed through the processors among the pool of computer hosts. This is performed via two different daemons that are native to the Linux operating system, namely RSH (Remote Shell) or SSH (Secure Shell).

The Windows operating system does have an equivalent RSH daemon that performs identical function. The SSH permits secure remote access over the network by negotiating and establishing an encrypted connection between the SSH client (Slave nodes) and the master node (server).

The RSH performs the same function as the SSH but it is far less secure, however it is a lot easier to setup and use; additionally, most CFD solvers come ready to use and accept instructions from the RSH. The clusters for this research were setup using the RSH as contained in the Red Hat (7.3) Linux distribution; further information may be obtained in Red Hat Linux user’s guide.

The MOSIX system is a management system that allows sharing of computational resources among a collection of Linux computers (x86 based nodes), including clusters, and a grid of dispersed computers within and across boundaries. The MOSIX software package turns networked computers running GNU/Linux into a cluster. It automatically balances the load between different nodes of the cluster and operates so that nodes can
join or leave the running cluster without disruption. The load is spread out among nodes according to their connection and CPU speeds. The advantage of the MOSIX system over the MPI based system is that there is no need to modify or link applications with any library or to assign processes to different nodes. The disadvantage is in the kernel dependency, in essence all the nodes must have the same kernel running all the Linux nodes, and this is not always practical.

5.5 Partitioning.

Partitioning is the process of dividing the mesh or grids into 'partitions' for the purpose of being solved simultaneously on the nodes making up the clusters. There are several methods for doing this in the literature, but most are based on the recursive bisection method, and only differ in the procedures. Consider the 2-dimensional planar deck section below (Fig. 5.4), the original mesh is first partitioned into two meshes of approximately equal size, the decomposition is then repeated recursively until the required number of partitions is accomplished.
5.6 Partitioning Methods.

There are essentially two methods of partitioning any 3-dimensional mesh namely:

- Element based
- Node based
In an element based method, the mesh partitioning is done along the element faces without dividing the element itself, i.e. at nodal locations. Node based partitions divide the mesh along the element faces i.e. between nodal locations as shown below in Fig 5.5

![Node based (left) and Element based(right) partitioning](image)

*Fig. 5.5 Node based (left) and Element based(right) partitioning*

The node based partitioning is used in the meshing and simulations for this research, since it is most consistent with the node based linear solver. The public domain program, *MeTiS* by Karypis & Kumar (1995), which uses a current and advanced multi level graph partitioning algorithm is used in the models for this research. The *MeTiS*, builds a graph containing the topology of the mesh to be partitioned and this graph is then coarsened down to a few hundred vertices. A bisection of the resulting much coarser graph is calculated and the resulting partitions are projected back onto the original graph, by consecutively refining the partitions. Although the *MeTiS* partitioner
algorithm is extremely fast and fully automated, it is however unable to take advantage of coordinate direction alignment, and a substantial amount of dynamically allocated memory is required to run it. This requires that the master node in the cluster be configured with additional memory for this operation, since all of this is done on the master node before being assigned to the slave nodes. As indicated in section 5.1, the computer nodes making up the cluster have different speeds; the ratio of the relative speeds is used by the partitioner to determine the size of the partitions. As an example, consider a parallel simulation involving two computers, one with a relative speed of 2.0 and the other with a relative speed of 1.0. The faster machine would work on a partition size twice as large as the slower machine in this case. However, for this research work the variation in computer speed (2.53-2.8 Ghz) is not that great, hence the sizes of the partition were in all cases almost identical.

An alternative to the MeTiS partitioner is the Recursive Coordinate Bisection partitioning algorithm as shown in figure 5.6, equally fast and fully automated but without the extra overhead of memory usage of the MeTiS. In this method, the partitioning is based on the global coordinates of the mesh, with the recursive bisectioning performed in the coordinate direction with the largest dimension. A particular disadvantage of this method is the probability that larger overlap regions can exist compared with the MeTiS partitioner, and each partition may contain separate parts.
5.7 Post-processing of results

The typical output from the combined steady state-transient simulation of a model is on average about 30 Gigabytes. The output file hold the mesh and the requested functions from the simulations such as pressure, vorticity, etc. Post processing in CFD analysis is best done using separate specialized software. For this research, the program Fieldview version 10 by Intelligent Light (www.ilight.com) was used. Fieldview can process the output from all the major CFD analysis software, though it rewrites the results in its own format. Thus a 30 Gigabyte output from say a CFX solver with 500 time steps, and a desired result at every other time steps, gives about a 275 Gigabyte
output from *Fieldview*. Manipulating and storing this amount of data for all the models and at different angles of attack requires a dedicated computer. Hence, the author had to purchase another computer for this purpose. For this dedicated post processing computer, each model has a four 250 Gigabyte hard drive for storage, a 300 Gigabyte hard drive for exporting the *Fieldview* post processing file, another 250 Gigabyte for back up and data over flow and a separate 200 Gigabyte for the operating system. Culminating in a 1.5 Terabyte system with 8 hard drives. These enormous data storage requirements and the associated costs imposed significant limits on the amount of modeling done in this research.

### 5.8 Summary

The massive size and the volume of the mesh that needs to be solved for the simulations for this research required the building of a computer cluster. This was accomplished by the author by researching parallel computing, computer architecture and hardware, memory management, alternative operating systems and clustering methodologies.

A functioning and functional cluster was built based on open source operating system and software using the Linux Redhat 7.3 distribution. This enabled the research to proceed, without which all the reported results after this chapter would not have been possible.
Numerical Modeling and Wind Tunnel Simulations.

6.0 Introduction

The complexities of wind flow introduced by the geometries of typical structures and by the characteristics of the terrain and obstacles upstream emphasize the need for detailed studies of wind tunnel models and simulation. This chapter describes the wind tunnel test(s) and results of the Carquinez Strait Bridge, and the equivalent numerical models used in this research.

The wind tunnel tests (Ragget et al, 1998) were performed by West Wind Laboratories for and on behalf of the State of California for the Office of Structures Design, California Department of Transportation. Wind tunnel tests have served as the basis of designing and obtaining aerodynamic forces (both static and dynamic) and related properties of bridges for quite a long time. Normally three types of tests are performed according to Scanlan & Simiu (1986):

- Tests on models of the full bridge. This entails a geometrically similar model of the full bridge while satisfying similarity requirements of mass distribution, reduced frequency, mechanical damping and shapes of vibration modes.
- Three-dimensional partial-bridge models. In this type of modeling, the main span or half of it is modeled as an economical approximation. These models are meant to simulate the vertical and torsional modes of response of the deck, which involves
the support of the deck section with taut wires or fine-wire catenary enveloped in a 3-dimensional simulated boundary-layer flow in the wind tunnel.

- **Section Model Tests.** These consist of the representative spanwise sections of the deck constructed to scale, supported by springs at the ends to allow for both vertical and torsional motion, while being enclosed between plates to reduce aerodynamic end effects.

Only the sectional and the scaled model test of the full bridge were performed on the Carquinez Strait Bridge, and only the section model test results involving the steady state lift, drag and moment coefficients are referred to in this research. Any other desired information beyond this is either not available or cannot be shared for this publication.

Numerical simulations are at best an attempt to duplicate an ideal wind tunnel test(s) using reasonable approximations and assumptions within the limitations of the existing turbulence models, available boundary conditions and the currently understood physics of flow simulations. The author reviewed and used several computational fluid dynamics programs, and found their use of boundary conditions and terminology to be similar, hence there is no need to refer to or endorse any specific software. The results of the simulations described here can and should be readily duplicated with a cross section of several solvers.
6.1 Details of the Carquinez Strait Bridge

Constructed in 2004, the Carquinez Strait Bridge is of the classical suspension type as shown in figure 6.1a, and it is the first suspension bridge constructed in the United States since 1937, it is also designed as one of the most aerodynamic structure of its kind. The Carquinez Strait is located approximately twenty miles Northeast of San-Francisco, and it carries the Sacramento River into the San-Francisco Bay. The new Carquinez Bridge spans the strait on California State Route (SR) 80. The total cost of the bridge (construction) is $500M (USD); engineering costs (including design, geotechnical and wind tunnel tests) are normally between 10-15% of the construction cost.

The overall span length of the bridge is 1056m, with the main span at 728m and two side spans of 181 m (North) and 147m (South) respectively and an overall edge to edge of deck width of 25.6m. The superstructure is made of stiffened steel box girder; the typical section is shown in figure 6.1b. Apart from the wind design criteria, the bridge crosses an active fault thereby necessitating further stringent seismic design criteria. The superstructure has a 2-lane carriageway (one lane for each direction) and a pedestrian walkway. Two vehicular traffic barriers railings separate the pedestrian walkway on the bridge, since the barrier are not identical in configuration, the bridge deck section aerodynamics should ideally be studied in winds from either side.
Fig 6.1a Elevation view of the Carquinez Strait Bridge-Details from CALTRANS.

Fig. 6.1b Typical Deck Section of the Carquinez Strait Bridge-Details from CALTRANS.
6.2 The Wind Tunnel Test

The wind tunnel studies were performed in a $1 \times 4$ m open return type atmospheric boundary layer wind tunnel. The tunnel was designed specifically for bridge section model and full-bridge model testing. Sketches of the wind tunnel are shown in figure 6.2a.

Fig. 6.2 $1 \times 4$ m Atmospheric Boundary Layer Wind Tunnel-(Ragget et al, 1998)

The wind tunnel extends 6.1 m upstream from the test section without flair or constriction. Atmospheric boundary layers are generated in this space with the use of
simulation spires and blocks on the wind tunnel floors. The test section is open without
walls or a ceiling and hence ambient pressure within the test chamber is essentially
constant. Furthermore, winds can flow around and over the models without constriction (as
in full-scale environment). Therefore, blockage effects are minimal, and wind speeds will
not be artificially accelerated around the model because there are no walls to constrict and
accelerate the flow.

The following instrumentation was used: section model displacements and force
transducer displacements (to measure aerodynamic forces on the section model) were
measured with Schaevits 050-HR LVDT Transducers (see Table 6.2 for properties) and
ATA-101 Analog Signal Conditioners. Mean wind speeds were measured with a Sierra
Instruments Model 618 Air Velocity Meter. Mean and fluctuating wind speeds were
measured with a total head tube and Setra System, Inc. 239 Pressure Transducer. Analog
signals from the transducers were digitized on a Keithley Metrabyte DAS-8 Analog-to-
Digital converter.

<table>
<thead>
<tr>
<th>HU Series Nominal Model Number</th>
<th>Linear Range Linearity (±% full range)</th>
<th>Sensitivity mV/v/s/in Per Lin</th>
<th>Impedance Impulses</th>
<th>Phase Shift</th>
</tr>
</thead>
<tbody>
<tr>
<td>050 HR</td>
<td>±0.032 ±1.27 0.10 0.25 0.25 0.50</td>
<td>5.6 220</td>
<td>430</td>
<td>4000</td>
</tr>
</tbody>
</table>

Table 6.2 Mechanical Properties of the 050-HR LVDT Transducers.
For sharp edged bluff bodies such as the cross section of the Carquinez Strait Bridge, static aerodynamic coefficient similitude between the model and the prototype requires that the two be only geometrically similar. Non-dimensional factors such as the Reynolds number are only required to be above a minimum threshold for the test to be valid. The section model shown in figure 6.3 was made to a scale of 1:50, 1.62 m in span, and was a rigid model of a portion of the prototype 81.82 m in span. The model was constructed of hard wood and Plexiglas (figure 6.5).

The model was a rigid model elastically supported at its four corners with springs (figure 6.4) attached to two rigid beams which in turn were hung from four, short, flexible cantilever beams. There was one rigid beam above the leading edge of the section model, and another above the trailing edge of the model. The rigid beams translate up and down in proportion to the loads at the leading and trailing edges respectively. The section model was also held from swaying in the wind with drag lines that were connected to flexible supports. The support translation was proportional to the drag force on the section model. The rigid beam and rigid frame translations were all measured with LVDT transducers. The LVDT output was then calibrated as a force transducer with known forces and weights.
Figure 6.3 Sectional Model of Carquinez Straits Bridge - (a) Top view (b) Bottom view-
Ragget (et al, 1998)
Fig. 6.4 Suspended sectional model of Carquinez Strait Bridge in Wind tunnel lab – Ragget

(et al, 1998)

Fig. 6.5 Plywood and Beam sectional model of Carquinez Strait Bridge.
6.3 The Study Models

The simulated models that are studied here are fixed, rigid bridge decks in three dimension with standard RANS CFD methods generally used in engineering applications.

For the studies described in this research there are four models identified below, including the Carquinez Strait bridge. The overall model dimensions are identical in all cases, the only modification is the part identified in figure 6.5 as edge shape.

Fig. 6.5a Model 1 - depicting as built & tested Carquinez Strait Deck Section.
Fig. 6.5b Model 2- Modified Edge Detail of Model 1 with Rounded Edges.
Fig. 6.5c Model 3- Modified Edge Details of Model 1 with Sharp Inclined Edges.
Fig. 6.5d Model 4 – Modified Edge Details of Model 1 with Oval Edge Details.
For the purpose of the discussions that follow, the above models are simply referred to as Model 1- Model 4. Model 3 is the typical sharp edged version of model 1; model 2 has a semi circular end, and model 4 is in the form of an oval.

6.4 Flow Parameters

Features of the flow over the deck sections are characterized by parameters such as lift force/coefficients, drag force/coefficient, Strouhal number, Reynolds number and moment coefficient. Each variable is defined as follows with reference to figure 6.6:

![Figure 6.6 Static Force Nomenclature & Wind Direction.](image-url)
Numerical Modeling and Solutions.
Chapter 6

\[ C_D = \frac{D}{0.5 \rho U^2 A} \]
\[ C_L = \frac{L}{0.5 \rho U^2 A} \]
\[ C_M = \frac{M}{U^2 B^3 A} \]
\[ R_e = \frac{R_e}{v} \]
\[ S_i = \frac{f_i l}{U} \]  \hspace{1cm} (6.1)

Where:

- \( C_D \) → Drag Coefficient.
- \( C_L \) → Lift Coefficient.
- \( C_M \) → Moment Coefficient
- \( U \) → Free Stream Velocity
- \( \rho \) → Density
- \( A \) → Projected Area (Cross section)
- \( B \) → Breadth of Section
- \( \nu \) → Kinematic Viscosity
- \( f_i \) → Frequency
- \( l \) → Length
- \( R_e \) → Reynolds Number
- \( S_i \) → Strouhal Number
In figure 6.6, the adopted conventions used in this report are as follows; the reference free stream velocity shown is referred to as west wind, the angle of attack is measured positive clockwise from the neutral axis and negative when it is counter clockwise. The moment $M$ as shown is positive clockwise.
6.5 Numerical Modeling & Boundary Conditions.

The numerical models in all the bridge sections studied consist of three main regions as shown in figure 6.7:

- The Fluid domain where the incompressible Navier-Stokes equations are to be solved based on the selected turbulent model
- The rigid or fixed structural domain consisting of the deck sections
- The interface region.

Domain dimension: Longitudinal Direction (Flow direction) = 3000 mm; Transverse Direction (Lateral) = 57.2mm and Height = 1000 mm

Fig. 6.7 Domain and boundary conditions.
The dimensional constraints of the domain are motivated by the wind tunnel size, the physics of the flow and available resources for solving the model problems. The height of the domain is 1000mm (height of the wind tunnel) or about $2 \times B$. The air flow induced by the fans in the wind tunnel is essentially two dimensional, this is modeled in figure 6.7, by this the two walls on the y-axis fit snugly such that there are no gaps between the built domain or control volume where the problems are solved and the outer wall of the deck section. The length of the domain is approximately $6 \times B$; this is based on recommendations in most CFD solvers as well as the practical and economic reality of the volume of cells that can be solved as discussed in chapter 4. The lateral (y-direction, fig. 6.7) dimension of the domain is 57.2 mm, the choice of this dimension is based on the desire to capture the effect of both the pedestrian and vehicular barriers on the left side of the sectional model as shown in figure 6.5a-6.5d. In figure 6.5a, the reader will be better able to appreciate these repeating posts and beam in the model atop the barrier(s). The numerical model considers only three posts as sufficient to capture the behavior, two posts, each adjacent to the boundary walls in the lateral direction and one in the middle; thus the lateral dimension was set.

The inlet boundary condition is defined in terms of the inlet velocity profile as measured during the wind tunnel test, the turbulence intensity, eddy length scale and the total pressure. Ragget et al (1998) report the horizontal mean velocity variation over the width of the bridge deck section model from the wind tunnel tests measured at the mid height of the tunnel as shown in figure 6.8. This is the inlet velocity profile used in all the simulations, scaled over the width of the domain. The fact that the width of the wind tunnel is more than the width of the computational virtual tunnel does not change the modeling in
any way, since the inlet flow is 2-dimensional. Depending on the solver used, this inlet velocity profile may be entered as a cloud of points (fig. 6.8b) or as a function derived by running a curve through the plotted velocity profile. In transient analysis, it is often necessary to 'kick start' the simulation with a perturbation of the inlet velocity in the orthogonal direction, a fluctuation of the velocity say \( 0.1 \frac{m}{s} \) is used to aid the initialization and formation of vortex shedding.

![Wind Tunnel Wall](image)

Fig. 6.8a Horizontal wind speed for inlet Velocity (\( \frac{m}{s} \)) profile-Ragget et al (1998)
The turbulence or eddy length scale, $l$, is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows. It is calibrated for pipes, but for external flows of the type here, it is usually taken as either 10% of the largest directional length of the domain or a fraction of the size of the object over which the flow is moving, the former seems more reasonable and it is adopted here as well.

Flows in large wind tunnels are rarely ever smooth (laminar) but can have extremely low levels of turbulence. Without spires and blocks used to generate turbulent atmospheric boundary layers, Ragget et al. (1998) reports that the ambient turbulence intensity (ratio of the standard deviation of the longitudinal velocity fluctuation to the mean velocity) in the wind tunnel is 3.39%. This is low compared to the target expression of 11.0% expected by the bridge designers at the bridge deck elevation. The assumption at the inlet for this research is 5% turbulence intensity, though this does not seem to have much significance in the results from the subsequent simulations.
In incompressible flow, the inlet total pressure and static pressure $p_t$ are related to the inlet velocity via Bernoulli’s equation:

$$p_{stat} = p_{stat} + \frac{1}{2} \rho (U \cdot U)$$

where $p_{stat}$ is the relative static pressure. The CFX-Solver solves for $p_{stat}$ in the flow field and is related to the absolute pressure $p_{abs} = p_{stat} + p_{ref}$. At the inlet $p_{stat} = 0$, so the only applied pressure is $\frac{1}{2} \rho U^2$. All pressures are measured relative to this reference value, which is used to avoid problems with round-off errors. These can occur when the dynamic pressures change in a fluid, which is what drives the flow, are small compared to the absolute pressure level. For example, low speed atmospheric air flow like as is experienced in a wind tunnel may have dynamic pressure changes of only a few Pascal or less, but the changes are relative to the atmospheric pressure of around 100,000 Pa.

Pressure outlet boundary conditions require the specification of static (gauge) pressure at the outlet. At the outlet the relative static pressure was set to zero.

The top, bottom and side surfaces of the rectangular domain are modeled using free slip wall boundary conditions. This means that:

- The shear stress is set to zero such that the fluid is not retarded, i.e.
  $$\tau_{wall} = 0$$

- The velocity normal to the wall is also set to zero.
  $$U_{n,wall} = 0$$

- The velocity parallel to the wall is calculated during the solution.
An alternative to the free slip boundary condition is the symmetry boundary condition. Traditionally, symmetry boundary condition is used when the physical geometry of interest and the expected pattern of the flow solution have mirror symmetry. An advantage to using the symmetry boundary condition for the parallel walls is the reduction of the computational mesh since the boundary is now placed along a plane of geometric and flow symmetry. They can also be used to model zero-shear slip walls in viscous flows. The quantities set to zero at the boundaries are:

- The normal component of velocity.
- Normal gradient of all other variables.

Using the symmetry boundary condition implies symmetry conditions for all equations.

The bridge deck section is modeled using a wall with the no-slip boundary condition. In the no-slip boundary, the properties of the flow adjacent to the wall-fluid boundary are used to calculate the shear stress on the fluid at the wall. Experiments and mathematical analysis have shown that the near wall region can be subdivided into two layers; a viscous sublayer, the innermost layer where the flow is dominated by viscosity. Further away from the wall, in the logarithmic layer, turbulence dominates this region. Finally, there is a region between the viscous sublayer and the logarithmic layer referred to as the “buffer layer”, where the effects of molecular viscosity and turbulence are both important. Figure 6.9 shows these subdivisions of the near wall region. If the assumed logarithmic profile of figure 6.9 reasonably approximates the velocity distribution near the wall, then it provides a means to compute the fluid shear stress at the wall as a function of the velocity at a given
distance from the wall. This is as a ‘wall function’ and the logarithmic nature gives rise to the well known ‘log law of the wall’.

![Subdivisions of Near-Wall Regions](image)

There are three methods for implementing the flow (no-slip) in the near wall region for turbulent flows:

- **Wall functions**, applied to cells immediately adjacent to a wall assuming this lie in the log-law region. This method employs special algebraic formulae to represent velocity, temperature, turbulence parameters, etc, within the boundary layer next to the wall. The major advantage of this method is that the shear layers near walls are usually modeled with relatively coarse meshes (fig. 6.10), thereby saving computational time and resources. Typically for this method, the first grid point is placed at $50 < y' < 500$ depending on the application, and the flow in the viscous sublayer and buffer layer does not have to be resolved.
The wall function approach in the CFX-Solver is an extension of the method of Launder and Spalding (1974). In the log-law region, the near wall tangential velocity is related to the wall-shear-stress, $\tau_w$, by means of a logarithmic relation. In the wall function approach, the viscosity affected sublayer region is resolved by using empirical formulas to provide near-wall boundary conditions for the mean flow and transport equations. The logarithmic relation for the near wall velocity is given by:

$$u^+ = \frac{U_t}{\kappa} = \frac{1}{\kappa} \ln(y^+) + C$$  \hspace{1cm} (6.2)$$

where $u^+$ is the near wall velocity, $\kappa$ is the von Karman constant, $C$ is a log-layer constant depending on wall roughness, $U_t$ is the known velocity tangent to the wall at a distance of $\Delta y$ from the wall and $y^+$ is the dimensionless distance from the wall. The problem with equation 6.2 is that it can become singular where the near wall velocity, $U_t$, approaches zero. An alternative velocity scale can be used instead of $u^+$ in the logarithmic region,
say $u^* = C^\mu k^{1/2}$, where $k$ is the turbulence kinetic energy and $C^\mu$ is turbulence model constant given as 0.09.

This scale has the useful property that is does not go to zero if $U_\tau$ goes to zero

By this definition the friction velocity can be restated as:

$$U_t \quad \frac{1}{\kappa} \ln (y^*) + C$$

The absolute value of the wall shear stress $\tau_w$, is then obtained from:

$$\tau_w = \rho u^* u_{\tau}$$

where:

$$y^* = \left( \frac{\rho u^* \Delta y}{\mu} \right)$$

The basic idea behind the scalable wall function which is peculiar to CFX is that it allows for a consistent mesh refinement that is independent of the Reynolds number of the application. The function limits the $y^*$ value used in the logarithmic formulation by a lower value of $y^* = \max(y^*, 11.06)$. 11.06 is the intersection between the logarithmic and the near wall profile. The computed $\bar{y}^*$ is therefore applied to the wall mesh and is not allowed to fall below this limit hence the terminology scalable wall function.

- **Two-layer models**, employed as combinations of high Reynolds number ($k-e$) model with a low Reynolds number or zero equation wall model. The latter is applied to the near wall region where the meshes are finely spaced as shown in figure 6.11 to resolve the viscous sub-layer.
- **Low Reynolds Number models**, in which the viscous effects are incorporated in the wall functions of the $k$ and $\varepsilon$ transport equations. The method resolves the details of the boundary layer profile by using very small mesh length scales in the direction normal to the wall. Turbulence models based on the $\omega$-equation such as the Shear Stress Transport (SST) model used here are suitable for this method. Note that the low-Reynolds number does not refer to the device Reynolds number, but to the local turbulent Reynolds number, which is low in the viscous sublayer.

Wall functions are appropriate when the walls can be considered as smooth, for rough walls, the logarithmic profile still exists, but the viscous sub-layer does not. A roughness height would have to be determined in this case. That is a function of an equivalent sand grain roughness. Wall friction depends not only on roughness height but also on the type of roughness. For the models here, the walls are considered to be smooth. Further details on
how to determine appropriate equivalent sand-grain roughness are found, for example in White (1979) and Schlichting (1979).

In Chapter 3, it was mentioned that one of the deficiencies of the $k-\varepsilon$ model is its inability to handle low local turbulent Reynolds number computations, as further discussed in this section. Complex damping functions can be added to the $k-\varepsilon$ model as well as the requirement of highly refined near-wall grid resolution ($y^+ < 0.2$) in an attempt to model low local turbulent Reynolds number flows. This approach often leads to numerical instability. Avoiding this issue is one of the claimed advantages of the $k-\omega$ based SST (Shear Stress Transport) model adopted for all the simulations in this research.

6.6 Steady State and Transient Flows.

The time dependence of the flow characteristics can be specified as either steady state or transient. Steady state simulations by definition are those mean characteristics do not change with time and whose steady conditions are assumed to have been reached after a relatively long time interval. However, there is a need to satisfy convergence criteria. In CFX, this is normally a measure of the local imbalance of each conservative control volume equation and is referred to as the residual. The CFX-Solver normally calculates solution to various equations given the pertinent boundary conditions for the model, during this process, an equation may not be satisfied exactly; the residual then is the value by which the right hand side of the equation differs from the left hand side. The recommended
residual per CFX for most engineering applications is $10^{-4}$ but for geometrically sensitive simulations, the recommendation is $10^{-5}$; a bluff body such as a bridge deck section would not be considered geometrically sensitive but an airfoil will qualify. Steady state simulations require no real time information to describe them. Many practical flows can usually be assumed to be steady after initial unsteady flow development.

Transient simulations require real time information to determine the time intervals at which the solver calculates the flow field. Transient behavior can be caused by the initially changing boundary conditions of the flow, as in start-up, or it can be inherently related to the flow characteristics, so that a steady state condition is never reached. Many flows do not have a steady state solution and may exhibit cyclic behavior such as vortex shedding.

Sometimes simulations which are run in steady state exhibit difficulty in converging and no matter what action is taken regarding mesh quality and time step size, the solution does not converge. This normally indicates transient behavior. A good way to test if a simulation is behaving or exhibiting transient behavior is to run a steady state simulation and see if the residual plots are oscillatory, then either by increasing or decreasing the time step size by some factor, re run the model. If the residual plot changes in proportion to the factor used in changing the time step, then the phenomenon is most likely a numerical effect. If the period stays the same, then it is probably a transient or cyclic effect.

In transient mode, a physical time step and the maximum number of iterations per time step must be specified. The proper selection of the time step and the number of sub-iterations within one time step are important factors that affects the speed of convergence and the numerical accuracy of the simulations. It is well known that taking a large time step
Numerical Modeling and Solutions.
Chapter

for the numerical simulation can save CPU time for transient cases but at a cost of numerical accuracy. The computation may even diverge if the time steps are too large for cases with complex geometries or cases with high Reynolds number. Increasing the outer-loop iteration can be a remedy for the loss of numerical accuracy resulting from the time step increase. However, using a large outer-loop iteration number will result in much longer CPU times. There is a trade-off between the time step length and outer-loop iteration numbers, in terms of the convergence speed and numerical accuracy.

For an unsteady solution, if accurate information during the whole transient process is expected from the numerical results then every time step must be converged to obtain sufficient accuracy for the whole process. The time step can be calculated from the CFL (Courant-Friedrichs-Levy) number according to Courant et al (1967) and Lax (1967). The physical meaning underlying this expression is that, the time step $\Delta t$, must be small enough to take account of variations of fluid flow due to the convection and diffusion effects produced in local regions (cells) of dimension $\Delta l$ within the domain length, $L$.

The CFL number is defined as:

$$ CFL = \frac{u \Delta t}{\Delta L} \quad (6.6) $$

Where $u$ is the characteristic free stream speed, $\Delta t$ is the time step, and $\Delta L$ is the size of the control volume. Theoretical study has shown that to get a stable simulation the largest CFL number anywhere in the flow field must strictly obey:

$$ CFL < CFL_{critical} $$

To optimize the time step, the optimization of CFL number is essential, which is both algorithm and problem dependent. Typical allowable values of $CFL_{critical}$ for simple, perfect
Numerical Modeling and Solutions.
Chapter 6

gas, viscous flow with implicit time integration according to Courant et al (1967) range from 0.1 to 1.2. For a transient case, if the information only at a certain time instant is needed, the results at former time steps do not necessarily have to be fully converged, provided that the final result is converged. If the iteration is stable, larger CFL number can be used, say 1 to 100, but this is case dependent. As the complexity of the flow conditions, geometry, and physical model increase, the maximum allowable CFL may be reduced.

For a fully implicit solver such as CFX, where the mass and momentum equation are solved in an implicit manner, then there is no requirement for the CFL number, in this case the CFL number may serve as a guide in determining a sufficiently appropriate time step to use in the simulation. If a small enough time step is taken and the solution within the time step converges, the global balances and RMS (Root-Mean-Square) of the residuals are low enough then it is safe to assume that the criterion $CFL < CFL_{critical}$ has been met. The residual is a measure of the local imbalance of each conservative control volume equation. It is the most important measure of convergence as it relates directly to whether the equations have been adequately solved. A convergence with a residual value between $1.0 \times 10^{-4}$ and $5.0 \times 10^{-4}$ is considered sufficient for most engineering applications, anything beyond this is largely of academic interest and will require double precision solution which implies more memory. Furthermore, if the convergence criterion within a time step is achieved, then it basically indicates that the time step(s) are adequately small enough that the equations governing the solution are been resolved satisfactorily.
6.6.1 Adopted Procedure

A practical procedure proposed and used here in this research is to initially solve the simulation using the steady state solver, and then a transient simulation is initiated using the output or the result of the steady state simulation as the starting or initialization of the transient simulation(s). This strategy refines the boundary conditions at the inlet for later use during the transient phase as its starting point. The characteristic solution of a steady state simulation does not contain time dependent terms. However, certain phenomenon of interest, such as vortex shedding, are time dependent and hence there is limited use here for steady state simulations. Unsteady flow solutions are time dependent and therefore more useful, where the transient are now related to the flow characteristics. When the simulation is performed over a long enough time, the response actually assumes some steadiness until it gradually decays and the resulting static forces reported are based on a time averaged values. Using an average velocity at the inlet and the domain length, it is simple dynamics to obtain the time for the fluid to travel the length of the domain, i.e. \( t = \frac{L}{u} \) where \( L \) is the domain length and \( u \) is the mean inlet velocity of air. In general, the fluid should go through the domain at least three times for the flow physics to be developed.

Depending on the turbulence model, the total time can now be divided to satisfy the model criteria. In this research, the time step(s) that meets both turbulence model and the residual target requirements is approximately \( \frac{L}{500} \), given that the average inlet velocity of the fluid is 3.3 \( m/s \) over a distance of 3m, the time of travel is 0.91 sec. If the fluid is to travel through the domain three times it will take 2.73 secs, the transient time steps is then obtained as \( \frac{2.73}{500} \) secs or 0.0055 secs.
6.7 Parametric Study on Variation of Turbulence Intensity.

It is customary in CFD simulations to use an inlet turbulence intensity of 5%, Ragget et al (1998) in the wind tunnel test report of the Carquinez Strait Bridge, measured a value of 3% as compared to a targeted value of 11%. Table 6.7 below shows the computed effect of variation of the turbulent intensity at the inlet on the static forces on the deck section. The range studied is typical of what might be expected in practice over open water.

<table>
<thead>
<tr>
<th>Model</th>
<th>Turbulent Intensity (%)</th>
<th>D (Drag) (N)</th>
<th>L (Lift)(N)</th>
<th>M (Moment)(N-mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.5</td>
<td>0.0155</td>
<td>-0.076</td>
<td>-0.0011</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>0.0156</td>
<td>-0.075</td>
<td>-0.0013</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>0.0162</td>
<td>-0.081</td>
<td>-0.0016</td>
</tr>
</tbody>
</table>

Table 6.7 Variation of Turbulent Intensity and Static Force.

From Table 6.7, it can be seen that halving and doubling the inlet Turbulent Intensity does not have much effect on the static forces; however as will be seen in Chapter 7, the wake flows and flow separation and recirculation are influenced. For the work reported here, the turbulent intensity assumed is at 5%.

207
6.8 Parametric Study on Variation of Turbulence Length Scale.

The turbulence length scale is a measure of the size of the energy containing large eddies in a turbulent flow. Two values have been used, 0.3 and 0.6 m, corresponding to 59% and 117% of the width of the deck. These scales are not untypical of the operating environment and of a size that may interact with flow development over the deck section. However, comparison of the results of calculations with the two values in (Table 6.8) shows no significant differences.

<table>
<thead>
<tr>
<th>Model</th>
<th>Turbulent Length Scale (% of Domain Length)</th>
<th>D (Drag)(N)</th>
<th>L (Lift)(N)</th>
<th>M (Moment)(N-mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10</td>
<td>0.0156</td>
<td>-0.075</td>
<td>-0.0013</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>0.0156</td>
<td>-0.075</td>
<td>-0.0013</td>
</tr>
</tbody>
</table>

Table 6.8 Variation of Turbulent Length Scale and Static Forces.
6.9 Summary

This chapter reviews the types of wind tunnel tests common to long span bridge deck sections. From the sectional wind tunnel type, the dimensions of the wind tunnel and the flow physics, a virtual wind tunnel encompassing the domain was built.

An inlet velocity profile measured from the mid-height of the wind tunnel and a turbulence intensity of 5% coupled with a length scale of about 10% of the longitudinal domain length are prescribed at the inlet of the domain. For solvers that require the $k$ and $\epsilon$ directly, a relationship exists in the form of: $k = \frac{3}{2} I^2 U^2$ where $I$ is the specified turbulence intensity.

$\epsilon$ can be approximated using: $\epsilon = \frac{k^{3/2}}{0.3D_h}$ where $D_h$ is the hydraulic diameter, for a rectangular domain $D_h$ can be replaced by the square root of the square of the length and breadth of the domain.

Wall function approximations are directly linked to the turbulent models for the no-slip boundary conditions. A scalable wall function model based on the two-layer model is ideal for implementing the no-slip boundary condition, but quite often they are coded specifically for a particular turbulent model but with a rather strict $y^+$ value criterion that are computationally expensive. The low Reynolds number wall function is suited for the $\omega$-based turbulent model such as Shear Stress Transport (SST) model adopted in chapter 3 as the preferred turbulence model for the simulations in the research.

The assumed conditions at the inlet regarding turbulence intensity and turbulent length scale were verified via parametric studies; the length scale variation has no effect whatsoever on the static forces, while there are minor but significant static force
differences between 5% and 10% turbulence intensity, and virtually none between 2.5% and 5%. A value of 5% was adopted and used for the work reported here.

Finally, a semi empirical approach is proposed in determining whether a steady state or transient simulation should be used for the simulation, when the residuals from the solution of the transport equations are sinusoidal, the flow physics are best resolved with a transient analysis. The choice of time steps is not only based on the CFL number but on the available resources in terms of CPU for the computations. A parametric study might be in order for fully implicit solvers, so that the appropriate time steps are selected.

A more practical solution is a hybrid steady state-transient method; where possible using the results of the steady solutions as the starting as the initial flow field for the transient simulations.

For the work discussed here, this is the methodology used. To capture the flow physics adequately, it is recommended that the fluid should go through the domain a minimum of three times as a recommendation based on experience.
On The Effects of Edge Details

7.0 Introduction

Bridge deck sections can be properly termed bluff bodies, as the separated shear layer generated at the leading edge plays an important role in the production of forces. The behavior of the shear layer separated from the windward edge and vortices shedding into the wake are dependent on the edge details. Therefore it should be expected that the drag coefficient, lift coefficient, moment coefficient and Strouhal number, which defines the periodicity of the vortex shedding will be influenced by these edge details.

Aerodynamic forces acting on a section are generated from the motion of the fluid surrounding the section. Therefore, it is necessary to know the kinematics of the fluid motion around the section in order to properly and appropriately evaluate its aerodynamic properties. The aerodynamic performance of bridge girders may be enhanced by appropriately shaped edge details, such enhancement(s) may be used to:

- Avoid or reduce the formation of coherent large scale vortices in the wake of the deck due to the relatively abrupt surface angles, and thus reduce vortex induced vibrations at low wind speeds.
- Modify the aerodynamically induced forces and moments experienced by the bridge.
- Improve the wind conditions over the bridge.
Increase the aerodynamic damping in torsion and thus enhance the critical wind speed for the onset of flutter.

For example, Wardlaw (1992) reports that enhancing the edge detail coupled with the use of open traffic barriers of the Lion's Gate Bridge in Canada increased the critical speed of the bridge deck section in smooth flow. Another example is the Longs Creek Bridge, which is a cable stayed bridge with a main span of 217 m between its towers. The deck section is shown in figure 7.0a.

![Fig. 7.0a As-built Details of Longs Creek Bridge – Wardlaw & Goettler (1968)](image)

Just like the Tacoma Narrows bridge, observed motion were noticed on the bridge with several amplitudes varying from a few centimeters to 20 cm on one particular occasion when the handrail was blocked with snow. Wardlaw & Goettler (1969) undertook extensive wind tunnel investigation of the phenomena that were observed on the bridge, adjudged to be as a result of vertical bending with a frequency of 0.6 Hz which took place in a wind speed of 40-50 km/h.

The wind tunnel tests confirmed that vortex shedding was responsible for the motion. In their subsequent wind tunnel tests, twenty two corrective modifications were evaluated. One of
these was the addition of fairings as shown in figure 7.0b, thereby modifying the edge detail. From this singular and simple act the amplitude of the motions was substantially reduced.

Fig. 7.0b Longs Creek Bridge: Effect of 2.4m fairings and soffit plate-Wardlaw & Goettler (1968).

In this chapter, the results of the numerical simulations are compared with the wind tunnel tests carried out for the Carquinez Strait Bridge. Hypothetical modified edges on this bridge will be used to study the effect of edge details. The chapter starts with an error analysis of the wind tunnel test results, the results are then modified to reflect this. Comparison of the wind tunnel test result is done with the numerical model of the same. The models with the modified edges are then compared to this original model (Model-1). The physics of vortex shedding is reviewed in light of the different edge details for the cases of bridge deck sections with and
without parapets and equipments. In the previous chapters on grids and turbulence modeling, several parametric studies were studied under various assumptions; here their flow physics are presented and compared.
7.1 Error Analysis and Corrections on the Wind Tunnel Test Results

All experiments or tests are subject to a certain amount of error, for most, the error(s) could be systematic/bias or random. The wind tunnel test report on the Carquinez Straight Bridge does not indicate any error analysis on the data or the instruments used for their measurements, hence prior to commencing the comparison of the wind tunnel test(s) results to the numerical simulations, it was necessary and important to examine the errors inherent in wind tunnel testing as this will allow better judgment between predictions and observation.

The correction quantities for lift, drag and moment are expressed as (Ishak et al, 2006):

\[ C_l = C_{lu} \frac{1 - \sigma}{(1 - \varepsilon_b)^2} \]  
\[ (7.1) \]

\[ C_d = C_{du} \frac{1 - \varepsilon_b}{(1 + \varepsilon_b)^2} \]  
\[ (7.2) \]

\[ C_m = C_{mu} + C_l \sigma \frac{(1 - \sigma)}{4 (1 - \varepsilon_b)^2} \]  
\[ (7.3) \]

where \( C_{mu} \) is the uncorrected moment coefficient, \( C_{du} \) is the uncorrected drag coefficient, \( C_{lu} \) is uncorrected lift coefficient, \( \sigma \) is a wind tunnel correction parameter given as:

\[ \frac{(\pi^2)}{48} \left( \frac{B}{h} \right)^2 \]

where \( B \) and \( h \) are breadth and height of the bridge deck section respectively.

Wind tunnel blockage is one type of problem than can introduce errors into a wind tunnel test. The physical presence of a model within a test section is known as solid blockage, which produces a decrease in the effective area. The body placed in a wind tunnel will partially obstruct the passage of air, causing the flow to accelerate and thereby increasing all
aerodynamic forces and moments at a given angle of attack. Solid blockage is a function of the
model size and test section dimensions. A common formula for correction of solid blockage is
given as (CALTRANS, 1999):
\[
\varepsilon_{sb} = \frac{K M_v}{A_n^{\frac{1}{2}}}
\]  
(7.4)
where \( K \) is a wind tunnel correction constant for solid blockage effects given as 0.74, \( M_v \) is
the model volume, and \( A_n \) is the test cross-sectional area. Another type of blockage effect is
known as wake blockage, which results from a velocity within the wake that is lower than the
free stream velocity. The effect of wake blockage is proportional to the wake size and thus to
the drag force that is measured on the model, and is given as:
\[
\varepsilon_{wb} = \left( \frac{B}{2h_t} \right) C_{du}
\]  
(7.5)
where \( B \) is the breadth of the bridge deck section, \( h_t \) is the height of the test section and \( C_{du} \) is
the uncorrected moment coefficient. Equation 7.5 is used to modify the velocity of the air
stream in the domain such that:
\[
V_c = V(1 + \varepsilon_{wb})
\]  
(7.6)
where \( V_c \) is the corrected velocity in the domain, and \( V \) is the inlet free stream velocity.

The measured quantities that must be corrected can be subdivided into two categories: stream
and model quantities. The most important stream quantity is the velocity at the model. The
model quantities of greatest interest are the lift, drag and moment coefficients that need to be
corrected in their non-dimensional forms.
Apart from these, additional errors may be termed as:

- Random errors
- Systematic or Bias errors

Random errors are relatively easy to deal with; as more data are obtained repeatedly the random errors are reduced, though not eliminated entirely. The minimization of random errors is a rather expensive undertaking. There were some rather obvious errors in the results from this particular wind tunnel test, as at certain angles of attack, the lift coefficients were miss-assigned in recording. While these were easily corrected, other more subtle errors may not be so easy to discern and treat.

At the root of the systematic errors during a wind tunnel test is the instrumentation used to measure the relevant variables of force, air density and air velocity. Most wind tunnel results are presented as force at a normalized wind speed. Therefore, the presented data is really a function of the tunnel determined product “$C_D \times A$” or “$C_L \times A$”, as shown in chapter 6, section 6.4 and from equation 6.1, these values are dependent on $F$, $\rho$ or $U$. Physical instruments must measure these quantities, and each of these instruments will make their own contribution to either the “$C_D A$” or “$C_L A$” term. An uncertainty analysis is a common exercise used to explore what is primarily responsible for the error of the measurement. In essence, the total uncertainty is the square-root of the sum of the squares of each of the individual instruments errors as shown below.

Drag:

$$\Delta C_D A = \sqrt{ \left( \frac{\partial C_D A}{\partial F_D} \Delta F_D \right)^2 + \left( \frac{\partial C_D A}{\partial \rho} \Delta \rho \right)^2 + \left( \frac{\partial C_D A}{\partial U} \Delta U \right)^2 }$$
In essence, the total uncertainty is the solution of the above equations that has all the partial derivative terms included. By applying the general uncertainty analysis presented in Coleman and Steele (1989) the uncertainties in the velocity, lift coefficient and drag coefficient were found in a relatively straightforward manner. The drag measurement error comes from three sources: accuracy of the data acquisition instruments, the repeatability of the measurements and spanwise variation in the downstream momentum deficit. Using the techniques in McGhee et al. (1988) and Coleman & Steele (1989), the uncertainties caused by the instruments and measurement repeatability were less than 1% and 1.5% respectively. Based on a statistical analysis of a representative low Reynolds number airfoil by Lyon et al (1997), the uncertainties caused by the spanwise variations are estimated at 3% for $Re=100000$ which is approximately in the range of the bridge decks analyzed. Overall uncertainty in the lift coefficient is estimated to be 1.5% or half of that proposed from the statistical analysis of Lyon et al (1997).
7.2 Comparison of Static Aerodynamic Characteristics

Bridge response to wind is primarily governed by the geometrical and aerodynamic properties of the deck cross section. Other properties such as mass, mass moment of inertia, eigenfrequencies and structural damping are important but are not necessarily determined via a wind tunnel test. However, the basic structural properties that form the basis of the aforementioned are compared in Table 7.2 below:

<table>
<thead>
<tr>
<th>Model</th>
<th>Area ((mm^2))</th>
<th>Moment of Inertia ((mm^4))</th>
<th>Radius of Gyration ((mm))</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(I_{xx})</td>
<td>(I_{yy})</td>
<td>(I_{zz})</td>
</tr>
<tr>
<td>1</td>
<td>136559.4</td>
<td>9.97×10^8</td>
<td>4.04×10^9</td>
</tr>
<tr>
<td>2</td>
<td>135906.7</td>
<td>9.87×10^8</td>
<td>4.01×10^9</td>
</tr>
<tr>
<td>3</td>
<td>134679.7</td>
<td>9.73×10^8</td>
<td>3.81×10^9</td>
</tr>
<tr>
<td>4</td>
<td>136964.4</td>
<td>10.×10^8</td>
<td>4.1×10^9</td>
</tr>
</tbody>
</table>

Table 7.2 Comparison of Structural Properties of Various Models.

From Table 7.2, it is evident that the structural properties of the models are virtually identical. This is of significant importance, in that the Carquinez Strait Bridge is located within an active seismic fault, and quite often as in this case, a dynamic analysis due to seismic load from an anticipated earthquake will have to be performed. As far as the United States (AASHTO) code is concerned, either the seismic load or the wind load may govern the structural design of a bridge or building. However, for the purpose of this research, only the effect of wind loading is of concern and that for structural behavior may be of importance and significance to the bridge engineer/designer, the models are fairly identical.
To investigate the 3-D flow physics around the models, three measurement planes were used. The planes are set to be parallel to the flow direction as shown in figure 7.1 and designated Y1, Y2 and Y3 respectively. The plane Y1 is at 1 mm from the face of the domain wall and is the midpoint of the first upright, the plane Y3 is halfway through the deck section, while the plane Y2 is in between Y1 and Y3 and a quarter through the deck section. These locations apply to the other three models as well, and hereafter the terms planes Y1, Y2, and Y3 will be used to refer to three planes as shown in figure 7.1

Fig. 7.1 Locations of measurement and study planes (Typical for all models).

Next, a comparison of the aerodynamic characteristics; firstly the drag coefficients, for the different models is made against the wind tunnel test results for both the corrected and uncorrected cases as shown in figure 7.2 and Table 7.3. The wind tunnel tests were performed up to ±12° incidence for both the west and east wind respectively. The numerical simulations were performed to ±8° for reasons of economy on computer time and software licenses.
West Wind – Drag Coefficients.

<table>
<thead>
<tr>
<th>Angle of attack, $\alpha$</th>
<th>Wind Tunnel Test$^1$</th>
<th>Wind Tunnel Test$^2$</th>
<th>Model -1</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-8^\circ$</td>
<td>0.123</td>
<td>0.111</td>
<td>0.105</td>
</tr>
<tr>
<td>$-6^\circ$</td>
<td>0.135</td>
<td>0.128</td>
<td>0.125</td>
</tr>
<tr>
<td>$-4^\circ$</td>
<td>0.148</td>
<td>0.140</td>
<td>0.148</td>
</tr>
<tr>
<td>$-2^\circ$</td>
<td>0.156</td>
<td>0.148</td>
<td>0.139</td>
</tr>
<tr>
<td>0</td>
<td>0.161</td>
<td>0.153</td>
<td>0.125</td>
</tr>
<tr>
<td>$+2^\circ$</td>
<td>0.172</td>
<td>0.163</td>
<td>0.125</td>
</tr>
<tr>
<td>$+4^\circ$</td>
<td>0.158</td>
<td>0.150</td>
<td>0.132</td>
</tr>
<tr>
<td>$+6^\circ$</td>
<td>0.150</td>
<td>0.143</td>
<td>0.149</td>
</tr>
<tr>
<td>$+8^\circ$</td>
<td>0.141</td>
<td>0.134</td>
<td>0.139</td>
</tr>
</tbody>
</table>

Table 7.3a Comparison of Wind Tunnel Test results and Numerical Simulations for drag.

$^1$ Uncorrected Wind Tunnel Test results  $^2$ Corrected Wind Tunnel Test results for errors.

Fig. 7.3 Plot of Table 7.3
**East Wind** – Drag Coefficients

<table>
<thead>
<tr>
<th>Angle of attack, $\alpha$</th>
<th>Wind Tunnel Test$^1$</th>
<th>Wind Tunnel Test$^2$</th>
<th>Model -1</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-8^\circ$</td>
<td>0.120</td>
<td>0.114</td>
<td>0.127</td>
</tr>
<tr>
<td>$-6^\circ$</td>
<td>0.140</td>
<td>0.133</td>
<td>0.135</td>
</tr>
<tr>
<td>$-4^\circ$</td>
<td>0.150</td>
<td>0.143</td>
<td>0.138</td>
</tr>
<tr>
<td>$-2^\circ$</td>
<td>0.153</td>
<td>0.145</td>
<td>0.142</td>
</tr>
<tr>
<td>0</td>
<td>0.143</td>
<td>0.136</td>
<td>0.129</td>
</tr>
<tr>
<td>$+2^\circ$</td>
<td>0.147</td>
<td>0.140</td>
<td>0.127</td>
</tr>
<tr>
<td>$+4^\circ$</td>
<td>0.138</td>
<td>0.131</td>
<td>0.130</td>
</tr>
<tr>
<td>$+6^\circ$</td>
<td>0.124</td>
<td>0.119</td>
<td>0.118</td>
</tr>
<tr>
<td>$+8^\circ$</td>
<td>0.111</td>
<td>0.105</td>
<td>0.104</td>
</tr>
</tbody>
</table>

*Table 7.3b Comparison of Wind Tunnel Test results and Numerical Simulations for drag.*

$^1$Uncorrected Wind Tunnel Test results  $^2$Corrected Wind Tunnel Test results for errors.

As can be seen from table 7.3, the drag coefficients obtained for model-1 compared very well with the wind tunnel test, especially when the latter had been corrected for the errors discussed in the previous section. Model-1 is essentially the same model that was tested in the wind tunnel and is therefore, a calibration based on the wind tunnel test. Models 2-4 can now be compared to Model-1, with the understanding that the same simulation criteria have been used. From table 7.3, it is evident that the discrepancy around 0° degree of angle of attack is largest. In this case, the viscous part of the calculated drag force is relatively low as a
proportion of the total force (CFX provides the viscous and the pressure contribution that sum up to the total force) as compared to the higher order degree of angle of attack, one possible explanation could be that the grid resolution on the deck surface in this particular instance is not totally capturing the viscous force, although in general grid resolution does not seem to have a significant influence on the overall response of the models studied here as will be shown later. The reason for this speculation is because it is generally considered important that a grid resolution be 'good enough' to capture the viscous sublayer off the deck surface. However, the difference between the calculated drag coefficients varies between 18-25% at 0 and 2 deg respectively when compared with the corrected (for errors) wind tunnel test results and in a way is significant. Efforts to refine the grid became prohibitively expensive, as the volume of cells required rapidly increased as the resolution was refined and it then became impractical to solve the problem with the computer resources available. As can be seen from the deck geometry, there are two barrier railings on the upwind side of the deck for a west wind flow, one of these barriers is meant to safeguard the pedestrian walkway, the elements that make for these railings are very small in geometry especially at the base. These smaller elements often determine the mesh level to be applied to the overall deck section. So then the two barriers are spanned by the deck section with fairly large surface areas and therefore a transition has to be made between the barriers and the deck surface, to accomplish this over a short distance is quite challenging and also might be responsible for some of the inaccuracies in the numerical simulations. As the angle of attack increases and the model is inclined, the deck geometry lends itself to smaller element sizes so as not to violate the cell geometry criteria such as low internal angle sufficient enough for solution, hence increased accuracy in capturing the viscous forces. At between 6 and 8 deg, the deck section is so positioned in the
domain that passage of the fluid over it is somewhat inhibited, since the contributing and resulting drag force comes from the contact of the air on the deck surface, this becomes difficult to accomplish effectively. One way to avoid this problem is to enlarge the domain considerably to accommodate a higher angle of attack so that vertical velocities at the upper and lower boundaries are insignificant. This would of course require much more computational effort. Another approach that some researchers have taken is to resolve the inlet velocity along the component axis and leave the object of study at 0 degree; i.e. have two inlet boundaries and two outlet boundaries. Resources were insufficient to investigate either strategy, so the possibility of some (apparently minor) error arising from the limitations due to domain size and choice of boundary conditions remains untested.

Fig. 7.2 Comparison of Drag Coefficients for all Simulated Models (West Wind).
From figure 7.2, it is evident that at low degrees of angle of attack (±2°) the effects of edge
details are minimal, beyond this point, model-2, with a round leading edge gave a lower drag
coefficient than the original model (Model-1) used for the Carquinez Narrows strait. Model-3
with the sharpest edged detail, gave the highest drag coefficient. While model-4 with an airfoil
like leading edge was marginally better. With the exception of Models 2 & 3, the drag
coefficients increased for the negative angle of attack; this phenomenon may have more to do
with the confluence of the effect of the edge detail and the east parapet that is fully exposed to
the fluid flow. For comparison purpose, consider the drag coefficient of both the replacement
of the Tacoma Narrows Bridge that collapsed and the New Burrard Inlet Crossing as shown
below in figure 7.3. Both bridges show increasing drag coefficients as the angle of attack
increases, a careful look at both results also shows that the wind tunnel tests were performed at
a higher rate of change in the angle of attack for each step, this undoubtedly has contributed as
the primary reason why prior research has failed to capture the aforementioned fact that at low
angles of attack the effect of edge details are not as critical as all the structures show similar
pattern of behavior. For both the New Burrard (Fig. 7.3b & 7.3c) and Tacoma Narrows (Fig.
7.3a), as the angle of attack increases, the drag coefficients also increases. This seems to
suggest that the sections may not be as efficient when compared with the models studied in this
research.
Fig. 7.3a Drag Coefficient of Tacoma Narrows (II)-Farquharson (1949-1954)

Fig. 7.3b Drag Coefficients New Burrard Inlet Crossing- R.L. Wardlaw (1970)

Fig. 7.3c Cross Section of the New Burrard Inlet Crossing- R.L. Wardlaw (1970)
The presence of parapets, barrier railings and equipments is a necessary safety issue for the protection of both vehicular and pedestrian traffic on the bridge. In this section, we consider the bridge models without any secondary structure on the deck models. Comparing figures 7.2 and 7.4, it can be seen that the presence of parapets and barriers increases the drag coefficients considerably (typically, from 0.06-0.08 for decks without parapets to 0.12-0.15 for those with it); secondly, the deck cross section’s drag efficiency are improved by the presence of these secondary elements. The behavior of the two sharp edged models (1 and 3) is markedly different without the parapets, with Model-3’s drag coefficient generally lower. The other two round edged models are virtually identical. It can then be deduced, that sharp edged deck sections are far more susceptible to variation drag forces than round edged deck sections. This is further reinforced by comparing Model-3 with Wardlaw’s (1970) New Burrard Crossing (fig. 7.3b)

Fig. 7.4 Drag Coefficients of Models without Parapets, Barriers and Equipments.
Lift Coefficients

Table 7.4 shows the comparison between the wind tunnel test results (corrected and uncorrected) and the numerical simulation results; both of which are remarkably in good agreement. Next, a comparison is made among the numerical models (figure 7.5) with their respective modified edge details, with and without the parapets and traffic barriers. In chapter two, galloping was defined as a one degree of freedom Aeroelastic phenomena, in which the excitation is produced by the movement of the member. According to Lawson (2001), for a bluff sharp edged body, the coefficient of the cross wind force can be written as:

\[
C_f(\alpha) = -C_L(\alpha) \cos \alpha - C_D(\alpha) \sin \alpha
\]

and if \( \alpha \) is small enough,

\[
C_f(\alpha) \approx -\alpha \left[ \frac{dC_L(\alpha)}{d\alpha} + C_D(\alpha) \right]
\]

for galloping to develop, the value of

\[
\frac{dC_L(\alpha)}{d\alpha} + C_D(\alpha) < 0
\]

this is the well known Den Hartog Criterion. However, its satisfaction does not always produce galloping. This criterion is significant in the understanding of the plots shown in figure 7.5. From figure 7.5a, it can be seen that the interaction of the effect of the modified edge details and the parapets are far more complex and leading to unstable response in the graph over the simulated angle of attack, particularly for Model-3 at -5° and 1° respectively. Model -1, representing the as-built Carquinez Strait Bridge, shows a fairly constant \( \frac{dC_L(\alpha)}{d\alpha} \) value with either an increasing or decreasing angle of attack. However, when the parapets are removed, Model-3 provides a smoother almost perfectly linear response. It is evident therefore, that there is a strong interaction between the type of parapets, barriers or similar equipments used on a bridge deck section and the edge detail.
On the Effect of Edge Details.

The angle at which the lift is zero also varies markedly for each of the models; for Model-1 it is approximately at 3.5° but for the sharp edge Model 3 there are two angles of attack at which the lift coefficient is approximately zero; suggesting a possible region of negative lift-curve slope and associated instability around 3-4° incidence. Figure 7.5b shows the same sets of models, albeit without the parapets. It is remarkable that the behavior of the Carquinez Strait model changes significantly, with zero lift now occurring at approximately 0.5°, a swing of 3° from the full model with parapets. In all cases the zero lift incidence is between 0 and 1° for the 'clean' decks, illustrating the importance of the parapets on the aerodynamics. The most notable change in the lift-curve is for Model 2, with the clean deck showing an almost constant lift-curve slope and associated stability, in contrast to the complex behaviour seen when parapets are included.

<table>
<thead>
<tr>
<th>Angle of attack, $\alpha$</th>
<th>Wind Tunnel Test$^1$</th>
<th>Wind Tunnel Test$^2$</th>
<th>Model -1</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-8^\circ$</td>
<td>-0.950</td>
<td>-0.917</td>
<td>-0.9380</td>
</tr>
<tr>
<td>$-6^\circ$</td>
<td>-0.722</td>
<td>-0.697</td>
<td>-0.7095</td>
</tr>
<tr>
<td>$-4^\circ$</td>
<td>-0.550</td>
<td>-0.531</td>
<td>-0.5594</td>
</tr>
<tr>
<td>$-2^\circ$</td>
<td>-0.338</td>
<td>-0.326</td>
<td>-0.3670</td>
</tr>
<tr>
<td>0</td>
<td>-0.188</td>
<td>-0.181</td>
<td>-0.1857</td>
</tr>
<tr>
<td>$+2^\circ$</td>
<td>-0.113</td>
<td>-0.109</td>
<td>-0.1116</td>
</tr>
<tr>
<td>$+4^\circ$</td>
<td>0.000</td>
<td>0</td>
<td>0.0247</td>
</tr>
<tr>
<td>$+6^\circ$</td>
<td>0.150</td>
<td>0.145</td>
<td>0.1801</td>
</tr>
<tr>
<td>$+8^\circ$</td>
<td>0.388</td>
<td>0.374</td>
<td>0.375</td>
</tr>
</tbody>
</table>

Table 7.4 Comparison of Wind Tunnel test (West Wind) results and Numerical Simulation.

$^1$Uncorrected Wind Tunnel Test results  $^2$Corrected Wind Tunnel Test results for errors.
The galloping motion and the associated potential instability are not present when the parapets are removed from the models. However, a comparison of models 1 and 2 reveals the significance of the influence that a parapet may have on the response of the deck section, clearly, the parapets used on the Carquinez Straits and their arrangements would not be suitable for usage on model 3 as their presence introduces instability. Secondly, it is further evident that the response of the bridge deck sections of the shapes described in this report (indeed any shape) can be modified with an appropriate parapet.

However, statutory agencies such as the California department of transportation that is responsible for safety on highways and bridges tend to have one or two types of parapets and barrier railings for general usage on all bridges and motorways. In essence, the barriers are
design and tested for structural safety rather than for aerodynamic considerations.

Nevertheless, the $dC_L(\alpha)/d\alpha$ of Model-2 is constant and therefore more stable without any parapet or barrier, and it will only require minimal effort to come up with the appropriate safety mechanism for it.

Figure 7.5a Lift Coefficients for Simulated Models with Parapets and Barrier Railings.
Figure 7.5b Lift Coefficients-West Wind without Parapets.

Figure 7.6a below shows the variation of the lift forcing function at 0° degree angle of attack. As air approaches and flows over the various deck sections, the effect of the different edge details and their influence on the lift force over time can be seen. Models 1 and 3 with the sharp edges have a fairly similar sinusoidal forcing function albeit with a lower lift force value, while the two round edge details shows the same similarity. Essentially, the lift forcing function consists of an initial steady state followed by a decaying transient function with variation at the onset time of the transient part respectively. This variation is clearly
attributable to the difference in the edge detail types. The decay rate of the transient part of the
lift forcing function is obviously related to the final lift force experienced by the different deck
sections. It can be argued that the edge detail enhances or reduces the lift force at the initial
steady state stage on impact from the wind. Again, as in the previous discussion on the lift
coefficients, a comparison is made without the presence of parapet and barrier on the deck
section. While the magnitude of the lifting force is different, as far as model 1 is concerned, the
form is the same.

![Graphs of lift force over time for different models](image)

Figure 7.6a Lift Force @ 0°: West Wind (With parapets)
Figure 7.6b Lift Force @ 0°: West Wind (Without parapets)
**West Wind** – Moment Coefficients.

In Table 7.5 below, the moment coefficients at various angles of attack are compared between the wind tunnel test and the geometrically identical numerical model 1. The numerical model shows fairly good agreements with the wind tunnel test. The corresponding moments used in calculating these coefficients were taken about the geometric center of the models.

<table>
<thead>
<tr>
<th>Angle of attack, $\alpha$</th>
<th>Wind Tunnel Test$^1$</th>
<th>Wind Tunnel Test$^2$</th>
<th>Model -1</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-8^\circ$</td>
<td>-0.0932</td>
<td>-0.0890</td>
<td>-0.097</td>
</tr>
<tr>
<td>$-6^\circ$</td>
<td>-0.0562</td>
<td>-0.0537</td>
<td>-0.058</td>
</tr>
<tr>
<td>$-4^\circ$</td>
<td>-0.0310</td>
<td>-0.0296</td>
<td>-0.0314</td>
</tr>
<tr>
<td>$-2^\circ$</td>
<td>-0.005</td>
<td>-0.00478</td>
<td>-0.00587</td>
</tr>
<tr>
<td>0</td>
<td>0.0174</td>
<td>0.0166</td>
<td>-0.0154</td>
</tr>
<tr>
<td>$+2^\circ$</td>
<td>0.0550</td>
<td>0.0525</td>
<td>0.0548</td>
</tr>
<tr>
<td>$+4^\circ$</td>
<td>0.100</td>
<td>0.0955</td>
<td>0.1005</td>
</tr>
<tr>
<td>$+6^\circ$</td>
<td>0.137</td>
<td>0.1308</td>
<td>0.1286</td>
</tr>
<tr>
<td>$+8^\circ$</td>
<td>0.15</td>
<td>0.143</td>
<td>0.1518</td>
</tr>
</tbody>
</table>

*Table 7.5 Comparison of Wind Tunnel test (West Wind) results of Moment Coefficient and Numerical Simulation.*
Figures 7.7a & 7.7b show the variation of the moment coefficients over the angles of attack of the simulation with and without the parapets. Twisting mode of motion is associated with the moment coefficients and their variation with incidences—when coupled with lift, gives the flutter derivatives as described in chapter 2. When the parapets are present, the various edge details exhibit a similar pattern of behavior i.e. a linearly varying moment coefficient over the various angles of attack studied. The notable differences are in the magnitude of the coefficient values at any angle of attack, and the rate of change of the moment coefficients with the angle of attack. Model 1 response is linear with constant slope regardless of whether the angle of attack is positive (when the soffit is exposed) or negative (when the top deck is exposed). When compared with Model 1, Models 2, 3 & 4 show lower moment coefficients and minor
rate of change when the angle of attack is negative. It is hard to say what might be responsible for the odd shapes in the graphs for models 3 & 4, perhaps it will be reasonable to draw a best fit line through the points for all models.

Fig. 7.7a Comparison of Moment Coefficients—With Parapets present on Deck.
Fig. 7.7b Comparison of Moment Coefficients—w/o Parapets present on Deck.
Pressure Distribution

The design of the superstructure is elemental; one part after another with differing stress criteria. For example the soffit plates have different thicknesses from the top plate, as a result of design code mandate. Hence in bridge design, the load combinations involving wind forces can be easily translated in terms of the stresses on each design element. In this section, a comparative study of the effect of the edge details on pressure distribution is presented for both with and without the parapets and barriers on the deck as shown in figures 7.8a-7.8w below. In all of the figures presented, the pressure variable is color coded instead of the vector length reflecting the pressure variation. This comes from CFX, while not ideal; its one must work with.
Fig. 7.8a (top) & 7.8b (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$. 

240
Fig. 7.8c (top) & 7.8d (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8e (top) & 7.8f (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8g (top) & 7.8h (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8i (top) & 7.8i (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8k (top) & 7.8l (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8m (top) & 7.8n (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
On the Effect of Edge Details.

Chapter 7

Fig. 7.8o (top) & 7.8p (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8q (top) & 7.8r (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8s (top) & 7.8t (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
Fig. 7.8u (top) & 7.8v (bot.) – Comparison of Pressure distribution on Deck sections
Fig. 7.8w (top) & 7.8x (bot.) – Comparison of Pressure distribution on Deck sections at $\alpha = 0^\circ$
From the above figures (7.8a-7.8x) we see that the immediate effect of the edge details on the deck sections is most significant at the inclined bottom flanges, the top deck plates immediately adjacent to the leading edge, around the parapets and at the trailing edge. Of course, the leading edge details show the maximum pressure levels, as would be expected. Models 2 and 4 with the rounded edges fared worst in the vicinity of the top deck plates, it appears then that there is a correlation between the exposed geometry of the deck section first encountered by the on-coming wind and how the pressure forces are subsequently distributed to other areas of the deck section.

Removing the parapets also leads to a significant drop and an improvement in the pressure distribution across the deck. It should be pointed out that the calculated pressure in vector form has been standardized from a base atmospheric pressure of 101321 Pa, this is normal practice, and also allow the post-processor to draw the vector representing the pressure distribution appropriately. However, the areas of negative pressure (suction) are not at first glance discernable; subtraction of the standard base atmospheric pressure would reveal this to be in the bottom flanges of the deck sections. The effect of the edge details appears to be minimal in these areas.
7.3 Mean Flow Distributions

Mean flow in each plane as indicated in figure 7.1 was obtained by the ensemble averaging of 500 instantaneous flow fields from the transient simulations. The following figures show the contour patterns and the corresponding velocity distribution.

Fig. 7.9a Contour & Velocity (m/s) distribution-Model 1(Center plane Y1)
Fig. 7.9b Contour & Velocity (m/s) distribution-Model 1 (Plane Y3)
On the Effect of Edge Details.
Chapter 7

Fig. 7.9c Contour & Velocity Distribution-Model 2(Center plane Y1)
On the Effect of Edge Details.

Chapter 7

Fig. 7.9d Contour & Velocity (m/s) Distribution-Model 2(PlaneY3)
Fig. 7.9e Contour & Velocity (m/s) Distribution-Model 3(PlaneY1)
On the Effect of Edge Details. 
Chapter 7

Recirculation regions

Reattachment point

Fig. 7.9f Contour & Velocity (m/s) Distribution-Model 3(PlaneY3)
On the Effect of Edge Details.
Chapter 7

Recirculation regions

Fig. 7.9g Contour & Velocity (m/s) Distribution-Model 4(Center plane Y1)
On the Effect of Edge Details.
Chapter 7

Recirculation regions

Reattachment point

Fig. 7.9h Contour & Velocity (m/s) Distribution-Model 4(PlaneY3)
Figures 7.9 (a-h) above compare the velocity flow regimes along two planes on the bridge deck from the numerical simulation— a plane through the mid deck section and one through the edge post, as shown in figure 7.1. The figures reveal the recirculation regions and mean reattachment points.

The recirculation regions are located in the same relative vicinity for all the different modeled sections. However, the flow speed of the air in the recirculation regions on the decks varies significantly. The region of the deck adjacent to leading edge in Model 1 shows a recirculation/eddy formation (fig. 7.9b) from the section through the parapets that is absent from the other models. Next, in the region between the posts that serves as the pedestrian walkway, the form and shape of the recirculating region differs between the models. This may have influence on pedestrian comfort in this region. The same could be said of the main carriageway. The posts create recirculation of the air but no vortex shedding is seen or, for that matter, a clearly defined reattachment at these locations. The variations in the velocity distribution show that both the sharp leading edges and rounded edges display similar 3-D effects. There are no noticeable Reynolds number effects as a consequence of the edge details in all the models studied, as the Reynolds number is fairly constant. Schewe (1998) reported that as the Reynolds number increases to about $3 \times 10^6$, there is a strong lift decrease however the Reynolds number values in these simulations were comparatively low (100000). Secondly, indirect Reynolds number effects are associated with changes in the pressure distribution arising from changes with Reynolds number in the boundary layer and in the wake development and these also were not observed. However, the flow regime around and on the deck section is highly three-dimensional.
7.4 Vortex Shedding

A vortex street is generally formed downstream from a bluff body such as a bridge deck section immersed in a moving fluid. If the structure is suspended elastically, it will undergo a form of oscillation induced by the periodic forces. The process of vortex shedding can only be explained when the effects of viscosity are considered, as only a viscous fluid will satisfy a no-slip boundary condition of its particles on a body surface (Fung, 1993). In general, the process of vortex shedding is dependent on the Reynolds number; this dependency is highly complex and therefore analytical and numerical simulations of the kind described in this research become rather challenging. The vortex shedding exerts a fluctuating force on the body and the body can consequently be excited to oscillate, Strouhal (1878) defined a dimensionless shedding frequency, the Strouhal number, to characterize this phenomenon:

\[ St = \frac{fd}{U} \]  

(7.1)

Where \( f \) is the shedding frequency and \( d \) is the across-flow dimension of the body. Since vortex shedding is such a complex phenomenon, many studies in the past have concentrated on the shedding from circular cylinders as with known behavior, this can be used as a calibration for numerical predictions. While this is instructive for academic purposes, the cylinder’s behavior is in no way comparable to a bridge deck section.

Matsumoto (1999) described various types and classes of bluff body vortex shedding; however, the type encountered in this research was not mentioned. This is a type of vortex shedding initiated in the wake of the deck section, particularly in the recirculation region off the trailing edge. On the inclined surface of the trailing edge the pressure is negative (suction)
and the velocity is also negative as shown below for the different models and various edge details examined in this research.

The recirculation region is not static, but rather dynamic and it is continuously changing in shape over time in the vicinity of the trailing edge, as the air flows over and under the deck.

Fig. 7.10a Recirculation at Trailing Edge of Model 1: Velocity (m/s) is shown in the legend-
measured in the recirculation region plotted.

With the exception of Model-3 with its pronounced sharp edge detail, all the other models show two clearly defined recirculation regions. While the precise locations of the regions are somewhat different, the effects are similar, with the notable exception of the width and 'volume' of the wake. The presence of three recirculation regions of model-3 does not pose
any significant change either in terms of the calculated Strouhal number or the frequency of the shedding.

It is the swirling of the air around the trailing edges that forces and alters the path of the air off the trailing edge, creating the vortex shedding. Significantly, there are large differences in velocity in the recirculating flow in the four cases, also in the maximum velocity in this region.

Fig. 7.10b Recirculation at Trailing Edge of Model 2: Velocity (m/s) is shown in the legend-measured in the recirculation region plotted.

The air speed in the flow around Model 1 varies between 0 and -1.204 m/s, for Model 2 between -0.179 to -1.259 m/s, Model 3 between -0.060 and -1.287 m/s and Model 4 between -0.157 and -1.424 m/s. The velocity in the recirculation region and the influence of the edge details is readily seen figures 7.10e-7h, when the full vortex street is considered. There is likely to be a direct relationship between the velocity in the recirculating region and any vortex
induced oscillations that the deck section may experience; the higher the speed in this region, the higher the propensity for vortex induced oscillations. This is because the force (pressure) is proportional to velocity squared and the rate as well as the magnitude at which the force is applied is related to the velocity. Hence, Model 4 may be most susceptible to vortex induced oscillations and Model 1 the least. The edge details determine (up or down) to a certain extent the initialization of the shedding as well as the distance measured from the edge of the deck in the wake to the extent of the recirculation region. In this regard Model-4 has the least, Models 1& 2 are about the same while Model 3 is longest, which coincidentally has the highest velocity in the recirculation zone. Figures 7.10e-7.10o shows the wavelength $L$, measured as a function of the depth of the deck $D$. The period is then $L/U$ and the frequency $U / L$; which is greatest for model 4. The advection velocity $U$, is constant for all the models.

Fig. 7.10c Recirculation at Trailing Edge of Model 3: Velocity (m/s) is shown in the legend—measured in the recirculation region plotted.
Consider figures 7.10i-7.10o, where the parapets and the barrier railings have now been removed. Here the shedding period \((L/U)\) for all the models have decreased significantly especially for Model 4 and hence the frequency \((U/L)\) has increased. Furthermore, the velocity in the wake is much higher when the parapets and barriers are not present. Hence, the parapets can be used to modify the response and behavior of the deck section. Increasing or changing the angle of attack does not change the frequency of the shedding; rather the effect of the edge details is seen more in the width of the wake. At 2.5 degree angle of attack the shedding begins to break up for Model 3 (figure 7.10n), for the other edge details, the velocity seems to be increasing in the wake region. Unlike the sections studied in Matsumoto (1999), and Deniz & Staubli (1998), the sections here do not have symmetry about their longitudinal axis. The soffit and the inclined webs of these models play a significant role in their response. In all these models, with or without parapets the velocity in the region of the soffit is different from that in the region of the top deck section. It is also interesting that the path carved by the vortex shedding in the wake region consistently has a lower velocity than in the surrounding regions and consequently a lower pressure, the pressure inside this wake region remains low as the flow separates and a net pressure force (pressure drag) is developed, for some models the velocity do eventually match that of the rest of the domain.
Fig. 7.10d Recirculation at Trailing Edge of Model 4 – Legend shows velocity in m/s measured in the recirculation region plotted.

Fig. 7.10e Vortex Shedding in the wake of Model 1 @ α=0
On the Effect of Edge Details.

Chapter 7

Fig. 7.10f Vortex Shedding in the wake of Model 2 @ \( \alpha=0 \)

Fig. 7.10g Vortex Shedding in the wake of Model 3 @ \( \alpha=0 \)

Fig. 7.10h Vortex Shedding in the wake of Model 4 @ \( \alpha=0 \)
Fig. 7.10i Vortex Shedding in the wake of Model 1 (w/o parapets) @ $\alpha=0$

Fig. 7.10j Vortex Shedding in the wake of Model 2 (w/o parapets) @ $\alpha=0$

Fig. 7.10k Vortex Shedding in the wake of Model 3 (w/o parapets) @ $\alpha=0$
On the Effect of Edge Details.
Chapter 7

Fig. 7.10l Vortex Shedding in the wake of Model 4 (w/o parapets) @ $\alpha = 0$

Fig. 7.10m Vortex Shedding in the wake of Model 2 @ $\alpha = +2.5$ – note small amplitude

Fig. 7.10n Vortex Shedding in the wake of Model 3 @ $\alpha = +2.5$
On the Effect of Edge Details.
Chapter 7

Fig. 7.10a Vortex Shedding in the wake of Model 4 @ $\alpha=+2.5$ – note small amplitude
7.5 Turbulent Kinetic Energy Distributions

One of the more important variables used in the study of turbulence and its evolution in the boundary layer is the turbulent kinetic energy (TKE). Assuming that the flow around the deck(s) can be partitioned into mean and turbulent parts, then the total kinetic energy of the flow is the sum of the kinetic energy of the mean and turbulent flows.

Turbulent kinetic energy (TKE per unit mass) is defined by the expression:

\[ TKE = \frac{1}{2} (u^2 + v^2 + w^2) \]

and often approximated as:

\[ TKE = \frac{3}{4} (u^2 + w^2) \]  \hspace{1cm} (7.6)

when data is insufficient for exact evaluation; i.e. in this case \( v^2 = \frac{1}{2} (u^2 + w^2) \) is assumed because \( v \) was not measured on the \( x-z \) measurement plane. This assumption is based on standard wind tunnel data for turbulent boundary layers. In turbulence intensity profiles of a shear layer, Arie (1956) show that three \( u, v \) and \( w \) directional components have a similar shape in distribution. The magnitude of the \( v \) component was also found to be somewhere between the \( u \) and the \( w \). The instantaneous values of TKE can vary dramatically, so it is often useful to calculate a mean value as depicted in equation (7.6). TKE is generated by eddies and can be suppressed by layers of air that become more stable.

Referring to figure 7.1 for the different study planes, below is the calculated turbulent kinetic energy from the numerical simulations of the various models. In these figures (11a-11h), contour format of the TKE are shown taken at different planes of the models to aid with visualization.
Fig. 11a: Model1(at 0 deg incidence) Kinetic Energy, units $m^2s^{-2}$. Plane Y3.
Fig. 11b: Model2 (at 0 deg incidence) Kinetic Energy, units $m^2 s^{-2}$. Plane Y3.
On the Effect of Edge Details.

Chapter 7

Fig. 11c: Model3(at 0 deg incidence) Kinetic Energy, units m²s⁻² -- Plane Y3.
On the Effect of Edge Details.

Chapter 7

Fig. 11d: Model4 (at 0 deg incidence) Kinetic Energy, units $m^2 s^{-2}$ - Plane Y3.
Fig. 11e: Model1 (at 0 deg incidence) Turbulent Kinetic Energy, units $m^2 s^{-2}$ - w/o parapets.
Fig. 11f: Model2(at 0 deg incidence) Turbulent Kinetic Energy, units m²s⁻²-w/o parapets.
Fig. 11g: Model3 (at 0 deg incidence) Turbulent Kinetic Energy, units $m^2s^{-2}$ - w/o parapets.
Fig. 11h: Model4(at 0 deg incidence) Turbulent Kinetic Energy, units $m^2s^{-2}$ - w/o parapets.
TKE is concentrated in the recirculation zones on the deck, at the trailing edge and in the wake region. When the parapets are removed from the simulation, Model-1 (Figure 11c) (which represents the deck section of the Carquinez Straits Bridge) shows the highest TKE value where the flow separates and the separated layer has a higher velocity gradient.

There are many cases where vortex induced oscillations predicted by wind tunnel tests are not observed in full scale bridges (Saito et al. 1999). Some researchers (Morgenthal, 2000 & Saito et al. 1999) suggest that these oscillations are caused by turbulence. However, it is recognized that turbulence controls the vortex induced oscillations of long span bridge sections, in this case, the natural vortex frequency are exacerbated by the gust of turbulence setting off uncontrollable catastrophic vibrations. In its simplest description, turbulence can be characterized by two parameters: Kinetic energy and a length scale. So then, the measured TKE on or off the section may serve as an indication of and a measure of the susceptibility of the deck section to vortex induced oscillations if the premise of turbulence as been the major controlling influence in vortex induced oscillations is accepted.

In this case, the TKE values do not change much between the four models while the parapets are present, although the values from the rounded edge models are smaller. The same pattern is seen when the parapets are removed.
7.6 Effect of Grid Types on Flow Distributions

In chapter 4, grid types and their influence on the solution methods were discussed. It was concluded that hexahedral elements produce more accurate results when compared with tetrahedrons. Now, an examination is made on the flow patterns exhibited by both grid types. It is generally known that incorrect results will be obtained when the predicted flow is influenced by the grid type, for instance an incorrect grid might induce premature separation from the deck or fake reattachment.

![Image: Fig. 12a- Tetrahedral Mesh, $\alpha=+2.5\text{deg.}$](image1)

![Image: Fig. 12b- Hexahedral Mesh, $\alpha=+2.5\text{deg.}$](image2)
It should be pointed out that in figure 4.15a (Chapter 4), the leading edge, around the parapets and in the trailing edge, the grid spacing is very fine, this is necessary and needed since the parapet elements have very small dimensions and therefore requiring that the mesh control criteria meets a percentage of their minimal dimension. The frequency of the vortex shedding in the wake of the tetrahedral based model is within 7% of the hexahedral model, an almost identical value to the static force parameters calculated in chapter 4. Furthermore, the velocity distributions are fairly close except in the recirculation region where the tetrahedral model is probably exaggerated, this is probably induced or caused by the fine meshing at the trailing edge where the flow shed from deck forms a free shear layer. As a consequence of this, the wake width of the vortex shedding is wider and more pronounced. In all, it appears that where for economic reasons or limitations in availability of grid generation software or for difficult geometry where tetrahedral mesh has to be used, it may be concluded that the practical information obtained from external flow problem such as this, are not that influenced by the grid type.
7.7 Parametric Study on the Effect of Certain Boundary Conditions on Simulated Results.

In setting up the simulations, turbulence intensity, length scale and grid volumes are some of the boundary condition variables that were discussed in chapter 4. In this section, the effects of the variables related to turbulence on the flow pattern of the deck section are examined. The two important variables are fractional intensity and the eddy length scale. Recall from chapter 6 on the numerical modeling, that the boundary conditions had the turbulent intensity at 5% and the length scale at 10% of the longitudinal domain length. Turbulence intensity is normally characterized as High, Medium and Low and is defined as:

\[ I = \frac{u'}{U} \]

where \( u' \) is the root-mean-square of the turbulent velocity and \( U \) is the mean velocity. External flows such as across cars, airplanes and bridge deck sections are generally considered to be low. These are usually between 1-5%.

Fig. 13a Inlet Turbulence Intensity of 2.5% @ 0 deg angle of attack.
Figure 13a above shows the vortex shedding in the wake of model 1 with half the turbulence intensity used in the numerical simulation reported earlier. There appears to be a decrease in the Strouhal number measured. Decreasing the turbulent intensity shortens the frequency of the vortex shedding. Not only that, the initialization of the vortex shedding occurs farther downstream when compared with fig. 7.10a. The recirculation region in the wake of the trailing edge is also less pronounced.

The turbulence length scale is a measure of the size of the large energy containing eddies in a turbulent flow. Turbulent length scale is normally a fraction of the domain length in the direction of the flow. For example in the standard k-epsilon model, it is generally estimated as:

\[ l = C_\mu \frac{k^3}{\varepsilon} \]

where \( C_\mu \) is a model constant with a value of 0.09.

Fig. 13b 5\% Turbulence length scale at 0 deg angle of attack.
Figures 13b & 13c shows the effect of halving and doubling the turbulence length scale. The simulation as described in chapter 6 uses 10% of the domain length or 0.3m, while figures 13b & 13c are at 0.15m and 0.6m respectively. It seems the turbulent length scale should be at a maximum the distance between the inlet and leading edge, and ideally half way between the inlet and leading edge. Longer length scale produces vortex shedding that takes longer to develop and at a reduced frequency.
7.8 Summary

This chapter compares the results of the numerical model of the Carquinez Straits with the Wind tunnel test(s). The computed static forces of Lift, Drag and Moment coefficients corrected for wind tunnel errors were in general good agreement with the numerical simulations. The calibrated Carquinez Straits models were then compared with three representative models with different edge details, with and without parapets and barrier railings. As far as the static loading is concerned, a rounded edge with the initially sloping edge detail (Model 2) provides the most stable results. However, the impact and influence of barrier and parapets are significant; their presence on a section can create stability or instability. This is significant as parapets are often replaced if and when it is damaged or sometimes if the loading criteria they are designed for changes. It is also determined here that the edge details have a patterned loading function. In this regard, the different edge details produce an initial transient part which quickly transits into an almost sinusoidal steady state. The decaying rates of these forcing functions are dependent on the edge details used.

As far as the static aerodynamic coefficients are concerned, the rounded edge Model 3 performed better while there are parapets present, it will be quite easy to design a parapet that will be compatible with it. Indeed, it is common nowadays to use an open restraint system to protect vehicular traffic; in that case the section will be excellent. The issue of vortex shedding and the oscillations that come with it has been of interest to researchers for quite a while. Most of it and its mechanics has been classified, however, in this research a form of vortex shedding, possibly peculiar to bridge deck section is identified. This type is generated and sustained by the recirculating air off the trailing edge aided by the inclined web that is so typical in long span bridge deck sections.
Conclusions and Future Research

8.0 Introduction

Long span bridge design requires expensive and often extensive wind tunnel testing to obtain important static and dynamic design parameters. A slight modification of any element of the deck section normally requires re-testing of such deck sections. The primary objective of this research is to investigate the effect of edge details on the aerodynamic characteristics and response of a long span bridge deck section. Since this objective will be accomplished numerically, an almost parallel objective of the research is to determine whether a computational strategy could be evolved using commercially available software with the appropriate turbulence model that would lead to a reduction in the number of such tests. To this end, the research modeled different edge detail sections that are common in the design of long span bridges. Among the problems faced by designers and engineers are the lack of a clear and concise procedure for the selection of appropriate edge detail for a long span bridge deck section, what the influence of such details are on the aerodynamic characteristics and the corresponding flow pattern, and a lack of concise and practical step by step procedure to model a deck section in 3-dimensional using currently available turbulence models.
Conclusions and Future Research
Chapter 8

To an extent it has been shown that the preliminary design of long span bridges can be economically achieved and “what if” scenarios that are common in the design process can be answered quite reasonably without resulting to expensive wind tunnel tests. The progress in the implementation of turbulence models in the CFD methodology in the last decade has been very promising. While this technique is useful and very reliable, it is still not able to replace wind tunnel testing completely for the final design, but acts rather as a complementary strategy and a means of reducing the number of wind tunnel tests. This research only concern one bridge and its wind tunnel test results, limiting as that may appear, it has been shown that with the appropriate RANS turbulence model, the appropriate mesh and computational setup, the static aerodynamic properties have been efficiently obtained, hence accomplishing the objective of the research. Also the effect of edge details can be clearly seen in the static properties especially when the basic deck section was studied without any parapets or equipment placed on the deck.

8.1 Contribution & Summary of the present study

Within the context of this research study, the presented body of knowledge from the numerical simulations on the effect of edge details on the aerodynamic characteristics of long span bridge deck sections can be summarized as follows:

- The use of Computational Fluid Dynamics with an (the) appropriate turbulence model as a form of virtual wind tunnel has been shown to be a realistic tool in the preliminary design of long span bridge deck sections.
- The Shear Stress Transport (SST) turbulent model proved to be the most appropriate among the typical Reynolds Averaged Navier Stokes (RANS) models.
examined for modeling external fluid flow normally encountered in the study of long span bridge deck section.

- Numerical accuracy was improved by use of appropriate grid type; while difficult to generate, hexahedral grids were found to be more accurate for models considered in this research when compared with tetrahedral elements.

- The primary purpose of the deck section model in the wind tunnel is to establish that the proposed geometry of the bridge section has aerodynamically stable characteristics. After establishing this, it is common to make modifications to the section to accommodate aesthetic requirements, addition of equipments that are essential for operation or any other constraints that the bridge might need to satisfy. Computational Fluid Dynamics has been shown to be a veritable means of studying and establishing the effects of such changes on the aerodynamic characteristics.

- In much previous research, the study of numerical flow simulations in bridge aerodynamics by previous research and researchers has been limited to a basic section in 2-D. The use of 2-D models is crude and limited at best. The present research demonstrated a modeling and solution strategy with the associated boundary conditions as a methodology for solving 3-dimensional problems, accounting for important structural elements such as parapets and barrier railings. Further more, these elements are shown to contribute to the stability and instability of the deck section depending on the edge detail used.
Conclusions and Future Research
Chapter 8

- Computational Fluid Dynamic (CFD) has been shown to be an invaluable tool in the evaluation of aerodynamic forces and flow visualization of external flows of this kind, permitting savings on expensive and time consuming flow visualization wind tunnel tests.

- The accuracy correlation with wind tunnel test results were remarkable and further validated CFD as a worthwhile and economical alternative to section model testing, allowing for the removal of inefficient sections quickly and the type selection and design of deck sections to carry on while formal testing of the final section is verified in a wind tunnel test.

- From the static force results, the rounded end with sloping edge (Model-2) was found to provide the most efficient section for the bridge studied.

- The frequency of the vortex shedding was not influenced by the edge details and by inference the Strouhal's number.

- Changes to parapets and barrier railings design if needed to be modified for structural or maintenance reasons should be accompanied by a clear study on their effects on the bridge deck section aerodynamic properties.

- Edge details influenced the lift forcing function.

- The mechanics of vortex shedding hitherto not discussed in the literature was identified; it develops from the recirculating air off the trailing edge at the inclined web of the deck section and associated pressure forces are identified.
8.2 Recommendations for future research

The numerical study of any subject implies computational solution at a high and refined level, this require fast, efficient computers with adequate support from technicians with the know how for the research to proceed smoothly. Therefore, future research in this area will require the appropriate tools as first step. Having these tools available, this research can proceed further with the following:

1. Flutter and the critical velocity are critical and important elements in the design of long span bridges. A flutter study would require that the deck section would be mounted on spring elements and be perturbed whilst in the air stream. This means that the mesh will have to deform, introducing a new (but novel) level of complexity. The fluid model (CFD) would have to be linked to a structural model (FEM). The combined use of the FEM and CFD programs (the latter solving for the deformed mesh) can then solve for the response of the deck section, allowing the flutter derivatives to be obtained and the critical velocity for the onset of flutter calculated.

2. The forcing load function for the different edge details shows a relative simple pattern, it may be possible to mathematically propose and solve equations perhaps for such loading functions for different types of common edge details that may be used for the design of long span bridges.

3. The inclined web of the deck sections seems to play a role in the recirculation of air at the trailing edge, and it may be worthwhile to study the relationship
between the angle of the inclined web and the recirculation and the vortex shedding.

4. The parapets and barriers on the deck when scaled have very small dimensions compared with the deck, yet their dimensions determine the meshing criteria. It is worthwhile to see if they can be replaced with a model with simplified geometry to simulate their behavior.

5. There is a need for further wind tunnel tests to determine whether the predicted aerodynamic forces and characteristic response and flow as predicted by the numerical simulations are accurate.

6. Further basic research is needed to develop faster grid based turbulence models to solve external flow problems such as described in this research. The end result would be to reduce the computational time from days to hours; admittedly faster computers will go a long way to resolve this issue.
8.3 CFD Modeling – Some Important Issues

While CFD offers tremendous potentials in solving the fluid-structure interaction problems encountered in bridge deck aerodynamics and its wider exploitation is therefore to be encouraged, it is important to recognize that effective application requires great care, physical insight and continuous validation by reference to experimental data. The large majority of experienced and seasoned Computational Fluid dynamicists will bear witness to having experienced frequent, frustrating and perplexing instances of numerical instability, agonizingly slow convergence, insufficient resolution with economically tolerable grid densities, and a high level of sensitivity to superficially minor boundary conditions and obvious lack of physical realism in the solutions generated. Figure 8.1 shows the interaction of some of the issues related to solving external flow problems using CFD.

![Fig. 8.1 Major CFD modeling issues.](image-url)
Conclusions and Future Research
Chapter 8

Here we look at some of these issues and the best practice for bridge deck analysis:

1. **Turbulence modeling.** Most flows of interest in practical engineering applications and interest are turbulent in nature and the turbulent mixing may then dominate the behavior of the fluid. This turbulent nature of the flow plays a crucial part in the determination of many of the important engineering parameters relevant to the design of bridges, such as lift, drag, moment coefficient, flow separation and vortex shedding. The turbulent states and modeling methods that are encountered in CFD applications are rich, complex and varied. For practical purposes, we are concerned only with the RANS based models, which can be roughly divided as:

   - Algebraic models (zero-equation)
   - One equation models
   - Two equation models
   - Stress Transport models

Since the models are based on different assumptions, all these models have limitations which depend on the modeling strategy. In general, the two equation model have wider applications and usage. Of these, the most popular is the $k-\varepsilon$, followed closely by the $k-\omega$ model. The performance of the two equation turbulence models deteriorates when the turbulence structure is no longer close to local equilibrium. This would occur when the ratio of production of turbulence energy to the rate at which it is dissipated at the small scales departs significantly from its 'equilibrium value' or equivalently when dimensionless strain...
rates become large. Various attempts have been made to modify the two equation turbulence models to account for strong non-equilibrium effects. Of these, the so-called SST (Shear Stress Transport) model is considered as one of the most robust and recommended here.

2. Grid generation. The computational grid represents the geometry of the subject of interest for the purpose of calculating the flow field. It consists of grid cells that must provide an adequate resolution of all the geometrical features. The grids must be fine enough to capture all important flow features. The accuracy of the simulation increases with increasing number of cells affording better resolution of the geometry. However, due to limitations imposed by the associated increased computer storage and run time some compromise is nearly always inevitable. In addition to the grid density, the quality of the mesh depends on various criteria such as aspect ratio, spatial distribution of cell sizes and distance of cell faces from boundaries among many others. Here are some guidelines that can be followed:

- Clean up the CAD geometry by ensuring that lines and surfaces are properly joined together, and for body fitted grids (recommended) check that the surface grid conforms to the geometry.

- Avoid the use of tetrahedral elements in the boundary layer but rather use hexahedron cells if at all possible.

- Avoid highly skewed cells, especially for hexahedral cells or prisms; the included angles between the grid lines should
Conclusions and Future Research

Chapter 8

be optimized in such a way that the angles are closest to 90 degrees.

- Away from the boundaries, ensure the cell aspect ratio is not too large; check the code criteria for this. This is most important near walls.

- The code requirements for mesh expansion ratios should be carefully observed. Mesh size discontinuities should be avoided and changes in mesh spacing should be smooth and continuous.

- Analyze the suitability of the mesh by a grid dependency study, where you use at least three different grid resolutions.

3. Boundary conditions. The boundary conditions necessary for a CFD simulation are surprisingly straightforward, however invoking and using the correct physics to describe what is intended can be difficult. In many applications, there is a frequent difficulty to define some of the boundary conditions at inlet and outlet of a calculation domain in the detail that is needed for an accurate simulation. A typical example is specification of turbulence properties (turbulence intensity and length scale) at the inlet flow boundary. In general, the boundary condition should closely adhere to the following:

- Use either a uniform inlet velocity or specified velocity profile.
• Inlet flow direction and speed should be consistent with the velocity profile. Averaging of the velocity profile may be good enough.

• The specification of the turbulent length scale, as an equivalent parameter for the dissipation \( \varepsilon \), is quite difficult. For external flows, a value of half the distance between the point of inlet and the object of interest seems appropriate. However, a parametric study will often be appropriate.

• For the specification of the turbulent kinetic energy \( k \), values should be used which are appropriate to the application. These values are generally specified through a turbulence intensity level. A value of 10% is found appropriate in this work and may be used for similar simulations in the absence of site specific data.

• The boundary condition imposed at the outlet should be selected to have a weak influence on the upstream flow. The most suitable outflow conditions are weak formulations involving specification of static pressure at the outlet plane.

• Be aware of the possibility of inlet flow inadvertently occurring at the outflow boundary, which may lead to
difficulties in obtaining a stable solution or even to an incorrect solution.

- Care should be taken on the boundary conditions imposed on solid wall are consistent with both the physical and numerical model used.
- If roughness on the wall is not negligible, significant levels of uncertainty can arise through incorrect specification of roughness within the wall functions.

4. **Wall functions.** Wall functions will nearly always be used to limit the finest cell resolution near surfaces and, hence, control the overall mesh size (i.e. the number of nodes). The near-wall region is not then explicitly resolved with the numerical model, but is bridged using so called wall functions. In order to construct these functions, the region close to the wall is characterized in terms of variables rendered dimensionless with respect to conditions at the wall. With the usual definitions, the dimensionless velocity, \( U^+ \) and dimensionless wall distance, \( y^+ \) are written as \( U/u_\tau \) and \( y^+ \rho u_\tau / \mu \) respectively. If the flow close to the wall is determined by conditions at the wall, then \( U^+ \) can be expected to be a universal function of \( y^+ \) up to some limiting value of \( y^+ \). This has been observed in practice, with a linear relationship between \( U^+ \) and \( y^+ \) in the viscous sublayer, and a logarithmic relationship, known as the law of the wall, in the layers adjacent to this so called log layer. These universal functions can be used to relate flow variables at the first computational mesh point at some distance \( y \) from the wall,
directly to the wall shear stress without resolving the structure in between. The only constraint on the value of $y$ is that $y^+$ at the mesh points remains within the validity of the wall functions. Wall function guidelines can be summarized as follows:

- The values of $y^+$ at the wall adjacent cells strongly influence the prediction of friction and hence drag. Thus particular care should be given to the placement of near-wall meshing.
- The meshing should be arranged so that the values of $y^+$ at all the wall adjacent mesh points are greater than 30 since the form usually assumed is not valid much below this value. It is advisable that the $y^+$ values do not exceed 100 and should certainly never be less than 11; this is mostly for the blending region. However, some codes account for this by switching to alternative functions if $y^+$ is <30 and one needs to be aware of this.
- Controlling $y^+$ values implies an iterative approach as it is a variable that can only be evaluated once a calculation has been undertaken.
- Cell centered numerical schemes have their integration points at different locations in a mesh cell than cell vertex schemes. Thus the $y^+$ value associated with a wall adjacent
cell differs according to which scheme is being used on the mesh. Care should be used when calculating the flow using different schemes or codes with wall functions on the same mesh.

5. **Near wall resolution.** Quite often a universal near-wall behavior over a practical range of $y^+$ may not be realizable everywhere in a flow. Under such circumstances the wall function concept breaks down and its use will lead and its use could lead to significant error, particularly if wall friction is important. The alternative is to fully resolve the flow structure through to the wall. Some turbulence models can be used for this purpose, but others are not capable of doing it. The Shear stress transport model is capable of this as it uses the standard $k-\varepsilon$ model in the interior of the flow and the $k-\omega$ model to resolve the wall region. Whatever modeling option is adopted, a large number of mesh points must be packed into a very narrow region adjacent to the wall in order to capture the variation of the flow variables. If resolving the flow to the wall without using wall functions:

- Make certain that the turbulence model being used is capable of resolving the flow structure through the wall.
- Use a stretching factor that is small enough for progressing the mesh spacing away from the wall (no more than a factor of 1.20 and used incrementally).
• The value of $y^+$ at the first node adjacent to the wall should be close to unity.

6. **Control Volume Setup.** This has to do with domain size, discretization and the advection scheme appropriate for the solution strategy. At a minimum, the depth of the domain should be twice the dimension of the object of study in the direction of flow, if the angle of attack is fairly high then three times the length in the direction of flow. The longitudinal dimension should be at least three times the dimension in the direction of flow, this will allow for the wake to be well developed for visualization. A combination of steady state and transient analysis is ideal where the software code for the analysis permits it, i.e. use the results of the steady solution as the starting point for the transient solution. Advection schemes are quite often code dependent and tied to the turbulence model. Review the implementation of the turbulence model in the software code as even the familiar ones are often implemented differently, then determine the appropriate scheme based on a combination of economy and accuracy for your use.

7. **Analysis of results, sensitivity studies and dealing with uncertainties.** Most commercial codes come with some kind of post-processing or can link to other software for such purposes. This allows many of the flow phenomena to be visualized or plotted in graphical form. Two important steps of post processing are to determine:

- Whether the result is sensible
- Whether the results are accurate.
Conclusions and Future Research

Chapter 8

Checking the reliability and accuracy of the solution may involve several steps such as reviewing conserved integral parameters such as the force/momentum balance for each time step against the tolerance level recommended for the application. Sensitivity studies should test the response of the solution to the choice of boundary conditions, the effect of viscous approximations or turbulence models and the use of different grid densities. The results of such test provide case specific guidance on steps that might be taken to improve the overall accuracy.


Garrick, I. E. (1938). On Some Reciprocal Relations in The Theory of Non-

Gimsing, N. J. (2000). *Cable Supported Bridges: Concept and Design*. John Wiley & Sons Ltd.


Larsen, A. (1998). "Advances in Aeroelastic Analysis of Suspension and Cable-


Reynolds numbers. 35th AIAA Aerospace Sciences Meeting, Reno, NV, AIAA.


Menter, F. R., M. Kuntz, et al. (2003). Ten Years of Industrial Experience with the SST Turbulence Model, CFX-5 Community Web Site.


Predicting Flutter Velocity of Bridge Sections." Journal of Wind Engineering and Industrial Aerodynamics 91: 291-305.


