Aerodynamics study of the flowfield at a shelterbelt

Chien-jung Yu
Iowa State University

Follow this and additional works at: https://lib.dr.iastate.edu/rtd

Part of the Aerospace Engineering Commons, Agriculture Commons, Atmospheric Sciences Commons, Bioresource and Agricultural Engineering Commons, and the Mechanical Engineering Commons

Recommended Citation
https://lib.dr.iastate.edu/rtd/11578
INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6” x 9” black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

UMI
A Bell & Howell Information Company
300 North Zeeb Road, Ann Arbor MI 48106-1346 USA
313/761-4700 800/521-0600
Aerodynamics study of the flowfield at a shelterbelt

by

Chien-jung Yu

A dissertation submitted to the graduate faculty
in partial fulfillment of the requirements for the degree of
DOCTOR OF PHILOSOPHY

Major: Aerospace Engineering

Major Professor: R. Ganesh Rajagopalan

Iowa State University
Ames, Iowa
1997

Copyright © Chien-jung Yu. 1997. All rights reserved.
Graduate College
Iowa State University

This is to certify that the Doctoral dissertation of

Chien-jung Yu

has met the dissertation requirements of Iowa State University

Signature was redacted for privacy.

Major Professor

Signature was redacted for privacy.

for the Major Program

Signature was redacted for privacy.

For the Graduate College
DEDICATION

This dissertation is dedicated to my parents. Chao-sen and Shei-shaw Yu. and my parents-in-law. Tsai-Chuan. Yu and Kuang-Kuang. Yuchiang. I also dedicate this work to my dear wife. Fenchin. and loving children. Deborah and Derek. Without their blessings. love. patience and support. none of this would have been possible. I give my deepest love and appreciation to all of them.
# TABLE OF CONTENTS

ACKNOWLEDGEMENTS ................................................. xiv

NOMENCLATURE ....................................................... xv

1 INTRODUCTION ..................................................... 1
   1.1 Background .................................................. 1
   1.2 Literature Review .......................................... 4
      1.2.1 Experimental Investigation .......................... 4
      1.2.2 Theoretical Calculation .............................. 8
      1.2.3 Summary .............................................. 11
   1.3 Present Research ........................................... 13

2 THEORETICAL BACKGROUND ....................................... 15
   2.1 Governing Equations ....................................... 15
   2.2 Control Volume Approach ................................... 16
      2.2.1 Integral Momentum Equations ....................... 16
      2.2.2 Integral Mass Equation ............................ 19

3 NUMERICAL ALGORITHM ......................................... 21
   3.1 Domain Discretization ..................................... 21
   3.2 Discretization Methodology ............................... 23
      3.2.1 Integral Governing Equations ....................... 23
      3.2.2 Discretization of Momentum Equations ............... 24
      3.2.3 Discretization of the Pressure Equation .......... 27
6.3 Shelterbelt Simulations .................................................. 104

7 CONCLUSIONS AND RECOMMENDATIONS .......................... 113

APPENDIX A ZERO-EQUATION TURBULENCE MODELS ............... 118

APPENDIX B FORMULATION OF HIGHER-ORDER ACCURACY SCHEMES .................................................. 123

APPENDIX C ROUGHNESS SPIRE DESIGN ................................. 139

APPENDIX D ENVIRONMENTAL WIND TUNNEL SETUP ............... 145

APPENDIX E COORDINATE TRANSFORMATION ........................ 150

APPENDIX F LINEARIZATION OF SOURCE TERMS DUE TO DRAG FORCES ................................................ 152

BIBLIOGRAPHY .................................................................. 155
LIST OF FIGURES

Figure 3.1 Staggered grid configuration ........................................ 22
Figure 3.2 Typical three-dimensional stencil .................................... 25
Figure 3.3 The near-wall nodes ...................................................... 39
Figure 3.4 Index system of one-dimensional grid ................................. 44
Figure 3.5 Relationship of Peclet number and $A(\|P\|)$ function [51] ...... 45
Figure 3.6 Flow chart for using SOUM scheme or QUICKM scheme ...... 46

Figure 4.1 Transect of the first field observation [67] (courtesy of Dr. Schmidt) 50
Figure 4.2 First field experiment setup ......................................... 51
Figure 4.3 Sketch of the observation mast ....................................... 52
Figure 4.4 Transect of the second field observation [68] (courtesy of Dr. Schmidt) 53
Figure 4.5 Second field experiment setup ...................................... 54
Figure 4.6 Positions of three observation masts from set No. 6 to set No. 10 of the first experiment [67] (courtesy of Dr. Schmidt) .......... 55
Figure 4.7 Vertical windspeed profile of each mast from set No. 6 to set No. 10 of the first experiment [67] (courtesy of Dr. Schmidt) ....... 56
Figure 4.8 Windspeed distribution and wind reduction percentage of set No. 21 of the second experiment ........................................ 57
Figure 4.9 Static pressure distribution across the shelterbelt of the second experiment [68] (courtesy of Dr. Schmidt) ......................... 58
Figure 4.10  Schematic diagram of wind direction across the shelterbelt . . . 59
Figure 4.11  Wind direction distributions in the shelterbelt flowfield, sets No. 60
1 and No. 55 from the second field experiment . . . . . . . . . . . . . . . . . . . . 60
Figure 4.12  ISU smoke tunnel (courtesy of Rockwell Collins) . . . . . . . . 62
Figure 4.13  Streamline contours surrounding the 20% porosity two-dimensional 63
model with 0 to 70 degrees approach angle . . . . . . . . . . . . . . . . . . . . . 63
Figure 4.14  Comparisons of the wake width at 0 degrees approach angle . . 64
Figure 4.15  Comparisons of the wake width at 10 degrees approach angle . . 64
Figure 4.16  Comparisons of the wake width at 30 degrees approach angle . . 65
Figure 4.17  Comparisons of the wake width at 50 degrees approach angle . . 65
Figure 4.18  Comparisons of the wake width at 70 degrees approach angle . . 66
Figure 4.19  Comparisons of the wake width at 0% porosity . . . . . . . . . . 66
Figure 4.20  Comparisons of the wake width at 10% porosity . . . . . . . . . 67
Figure 4.21  Comparisons of the wake width at 20% porosity . . . . . . . . . 67
Figure 4.22  Comparisons of the wake width at 30% porosity . . . . . . . . . 68
Figure 4.23  Comparisons of the wake width at 40% porosity . . . . . . . . . 68
Figure 4.24  Comparisons of the wake width at 50% porosity . . . . . . . . . 69
Figure 4.25  Comparisons of the wake width at 60% porosity . . . . . . . . . 69

Figure 5.1  Physical domain of a two-dimensional driven-cavity flow problem 73
Figure 5.2  Computational domain of a two-dimensional driven-cavity flow 73
with a 22 by 22 non-uniform grid . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 73
Figure 5.3  U-velocity profile on the vertical centerline of the driven-cavity 74
at Re=400 with a 22 by 22 non-uniform grid . . . . . . . . . . . . . . . . . . . . . 74
Figure 5.4  U-velocity profile on the vertical centerline of the driven-cavity 74
at Re=1000 with a 22 by 22 non-uniform grid . . . . . . . . . . . . . . . . . . . . . 74
Figure 5.5 Convergence history at Re=400 with a 22 by 22 non-uniform grid for the driven-cavity flow ................................. 75
Figure 5.6 Convergence history at Re=1000 with a 22 by 22 non-uniform grid for the driven-cavity flow ................................. 75
Figure 5.7 Physical domain of a two-dimensional backward-facing step ... 77
Figure 5.8 Computational domain of a two-dimensional backward-facing step 78
Figure 5.9 Streamline patterns of a backward-facing step for Re=200, 400, 600 and 800 ......................................................... 79
Figure 5.10 Dimension of the first separation region for the backward-facing step flow ......................................................... 80
Figure 5.11 Dimension of the secondary separation region for the backward-facing step flow ......................................................... 80
Figure 5.12 Convergence history at Re=200 for the backward-facing step flow 81
Figure 5.13 Convergence history at Re=400 for the backward-facing step flow 81
Figure 5.14 Convergence history at Re=600 for the backward-facing step flow 82
Figure 5.15 Convergence history at Re=800 with a 102 by 102 non-uniform grid spacing for a backward-facing step flow ..................... 82
Figure 5.16 U-velocity profile at x=7 channel heights for a backward-facing step flow at Re=800 ......................................................... 83
Figure 5.17 U-velocity profile at x=15 channel heights for a backward-facing step flow at Re=800 ......................................................... 83
Figure 5.18 Physical domain of the flat-plate flow ................................. 85
Figure 5.19 Numerical domain of the flat-plate flow ................................. 85
Figure 5.20 Velocity profile comparison of the flat-plate flow at Re=10000 86
Figure 5.21 Convergence history of the flat-plate flow at Re=10000 .................. 86
Figure 5.22 Physical domain of the channel flow ................................. 88
Figure 5.23 Numerical domain of the channel flow ................................. 88
Figure 5.24 Velocity Profile comparison of the channel flow: x=20 channel heights (at exit) and Re=1000 ........................................ 89
Figure 5.25 Convergence history of the channel flow at Re=1000 .......... 89
Figure 5.26 Physical domain of the 2-D block flow .......................... 91
Figure 5.27 Numerical domain of the 2-D block flow ......................... 91
Figure 5.28 Streamline contours of the 2-D block flow for Re=50, 100, 1000, 10000 and 100000 .................................................. 92
Figure 5.29 Physical domain of a three-dimensional cavity flow .......... 94
Figure 5.30 Grid distribution on the x-y plane of the three-dimensional cavity flow .............................................................. 94
Figure 5.31 U-velocity profile along the vertical centerline on the xy-center plane at Re=100 for a three-dimensional cavity flow ............ 95
Figure 5.32 U-velocity profile along the vertical centerline on the xy-center plane at Re=400 for a three-dimensional cavity flow ............ 95
Figure 5.33 U-velocity profile along the vertical centerline on the xy-center plane at Re=1000 for a three-dimensional cavity flow ......... 96
Figure 5.34 Comparison of convergence history at Re=1000 for a three-dimensional cavity flow ........................................... 96
Figure 5.35 Physical domain of a three-dimensional duct flow .......... 98
Figure 5.36 Grid distribution on the x-y plane of the three-dimensional duct flow .............................................................. 99
Figure 5.37 Horizontal velocity profiles on the xy-centerplane at the station of 30 duct-widths for a fully developed duct flow at Re=1000 .. 99
Figure 6.1 Physical domain of the shelterbelt flow .......................... 106
Figure 6.2 Numerical domain of the shelterbelt flow ........................ 106
Figure 6.3  Comparisons of numerical and observed vertical profiles of normalized horizontal windspeed at $z/H = 4.2$ downwind  

Figure 6.4  Comparisons of numerical and observed horizontal profiles of normalized horizontal windspeed at $z/H=0.4$ and $z/H=1.9$  

Figure 6.5  Numerical vertical profiles of normalized horizontal windspeed at different downwind locations  

Figure 6.6  Relationship between the drag coefficient normalized by freestream dynamic pressure and the solidity of shelterbelt  

Figure 6.7  Streamline contours of different drag coefficients normalized by freestream dynamic pressure: $C_d=0.15, 0.19, 0.27, 0.35, 0.45, 0.75$. and a solid shelterbelt from bottom to top  

Figure 6.8  Relationship between the drag coefficient normalized by freestream dynamic pressure and the drag coefficient normalized by local dynamic pressure  

Figure A.1  Two-layer edge viscosity model  

Figure B.1  One-dimensional grid format  

Figure B.2  U-velocity profile on the vertical centerline of the driven-cavity at $Re=3200$ with a 22 by 22 non-uniform grid  

Figure B.3  U-velocity profile on the vertical centerline of the driven-cavity at $Re=400$ with a 42 by 42 non-uniform grid  

Figure B.4  U-velocity profile on the vertical centerline of the driven-cavity at $Re=1000$ with a 42 by 42 non-uniform grid  

Figure B.5  U-velocity profile on the vertical centerline of the driven-cavity at $Re=3200$ with a 42 by 42 non-uniform grid  

Figure B.6  U-velocity profile on the vertical centerline of the driven-cavity at $Re=400$ with a 82 by 82 non-uniform grid
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>B.7</td>
<td>U-velocity profile on the vertical centerline of the driven-cavity at Re=1000 with a 82 by 82 non-uniform grid</td>
<td>134</td>
</tr>
<tr>
<td>B.8</td>
<td>U-velocity profile on the vertical centerline of the driven-cavity at Re=3200 with a 82 by 82 non-uniform grid</td>
<td>135</td>
</tr>
<tr>
<td>B.9</td>
<td>Convergence history at Re=3200 with a 22 by 22 non-uniform grid for the driven-cavity flow</td>
<td>135</td>
</tr>
<tr>
<td>B.10</td>
<td>Convergence history at Re=400 with a 42 by 42 non-uniform grid for the driven-cavity flow</td>
<td>136</td>
</tr>
<tr>
<td>B.11</td>
<td>Convergence history at Re=1000 with a 42 by 42 non-uniform grid for the driven-cavity flow</td>
<td>136</td>
</tr>
<tr>
<td>B.12</td>
<td>Convergence history at Re=3200 with a 42 by 42 non-uniform grid for the driven-cavity flow</td>
<td>137</td>
</tr>
<tr>
<td>B.13</td>
<td>Convergence history at Re=400 with a 82 by 82 non-uniform grid for the driven-cavity flow</td>
<td>137</td>
</tr>
<tr>
<td>B.14</td>
<td>Convergence history at Re=1000 with a 82 by 82 non-uniform grid for the driven-cavity flow</td>
<td>138</td>
</tr>
<tr>
<td>B.15</td>
<td>Convergence history at Re=3200 with a 82 by 82 non-uniform grid for the driven-cavity flow</td>
<td>138</td>
</tr>
<tr>
<td>C.1</td>
<td>Spires and roughness in a rectangular test section [78]</td>
<td>141</td>
</tr>
<tr>
<td>C.2</td>
<td>Spires and roughness blocks installed in ISU environmental wind tunnel [79] (courtesy of Mr. Thernelius)</td>
<td>144</td>
</tr>
<tr>
<td>D.1</td>
<td>ISU environmental wind tunnel setup</td>
<td>147</td>
</tr>
<tr>
<td>D.2</td>
<td>Wiring diagram of ISU environmental wind tunnel for shelterbelt analysis</td>
<td>148</td>
</tr>
<tr>
<td>D.3</td>
<td>Calibration curve of a potentiometer</td>
<td>149</td>
</tr>
<tr>
<td>D.4</td>
<td>Calibration curve of a voltmeter</td>
<td>149</td>
</tr>
</tbody>
</table>
Figure E.1  Relationship between two frames  

Figure F.1  Two-dimensional shelterbelt flow cases
ACKNOWLEDGEMENTS

To my major professor, Dr. R. Ganesh Rajagopalan. I wish to express my deepest appreciation and gratitude for his suggestions, patience, support and guidance throughout the course of this research. By his gentle but persuasive criticism, he has molded me into a better student, researcher and teacher. For all of these, I am grateful.

I thank Dr. Gene S. Takle for inviting me to join the shelterbelt field observation research group. His timely discussions and suggestions were very helpful in my research.

I appreciate Drs. James D. Iversen, Jerald M. Vogel, William D. James and Robert J. Gregorac willingness to serve on my committee and to take the time to read through my dissertation.

I wish to thank my officemates and colleagues for making Room 116 a nice working place.

I thank the department secretaries, Gloria. Colleen. Dee. Laurie and Sue for their great help and kind caring, not only of my teaching career, but also of my research and of my family.

I would also like to thank Mr. Guo-Rong. Hu whose encouragement and mental support really kept me from feeling alone in this endeavor, and in particular, our common language allowed us to communicate with each other without any hindrance. I sincerely appreciate this.

With the grace, blessings and love from God Almighty. I am able to not only enjoy this wonderful journey in the scientific world, but also to ensure the way to eternal life. Hallelujah. praise the Lord!
NOMENCLATURE

Roman Symbols

\( A \) = area

\( a \) = coefficient of discretized governing equation

\( b \) = source term of discretized governing equation

\( C_1, C_2, C_\mu \) = constants used in TKE-\( \epsilon \) model

\( C_\nu, C_\epsilon \) = constants used in Prandtl-Kolomogorov expression

\( C_d \) = normal pressure drag coefficient

\( C_f \) = skin-friction drag coefficient

\( D \) = diffusive conductance

\( \vec{e}_r, \vec{e}_\theta, \vec{e}_\phi \) = unit vectors in the local spherical system

\( F \) = mass flow rate

\( \vec{F}_c \) = convective flux (vector)

\( \vec{F}_d \) = diffusive flux (vector)

\( G \) = constant used in TKE-\( \epsilon \) model

\( H \) = height of shelterbelt

\( h \) = height of channel

\( \vec{i}, \vec{j}, \vec{k} \) = unit vectors in the global system
General symbols:

- \( \bar{J} \) = general total flux (vector)
- \( k \) = turbulent kinetic energy
- \( L \) = characteristic length
- \( l \) = characteristic length
- \( P \) = Peclet number
- \( p \) = pressure
- \( Re \) = Reynolds number
- \( S \) = general source term
- \( t \) = time
- \( U_{inf} \) = inlet velocity
- \( u, v, w \) = velocity components in the global system
- \( \vec{v} \) = velocity (vector)
- \( V_{tot} \) = total velocity
- \( V_{inf} \) = freestream velocity
- \( \vec{w} \) = vorticity vector

Greek Symbols:

- \( \Gamma \) = diffusive coefficient
- \( \Phi \) = general dependent variable
- \( \alpha, \beta \) = geometric constants
- \( \alpha, \beta \) = Euler angles
- \( \delta \) = geometric constant
- \( \delta \) = boundary layer thickness
- \( \epsilon \) = turbulent dissipation
- \( \gamma \) = geometric constants
- \( \kappa \) = Von Karman constant
\[ \mu \quad = \text{dynamic viscosity} \]
\[ \mu_{eff} \quad = \text{effective viscosity} \]
\[ \nu \quad = \text{kinematic viscosity} \]
\[ \sigma \quad = \text{porosity of the shelterbelt} \]
\[ \rho \quad = \text{density} \]
\[ \sigma_s, \sigma_k \quad = \text{constants used in TKE-\(\epsilon\) model} \]
\[ \tilde{\tau} \quad = \text{shear stress tensor} \]

**Subscripts**

- \( E, W, N, S, T, B \) = east, west, north, south, top, bottom and central primary grids
- \( e, w, n, s, t, b \) = east, west, north, south, top and bottom auxiliary grids
- \( l \) = laminar condition
- \( nb \) = neighboring grid point
- \( p \) = quantity used in pressure equation
- \( p' \) = quantity used in pressure correction equation
- \( t \) = turbulent condition
- \( u, v, w \) = quantity used in \( u, v, w \)-momentum equations
- \( x, y, z \) = Cartesian coordinate system (global system)

**Superscripts**

- \( \ast \) = calculated quantity
- \( \prime \) = correction of quantity
- \( 0 \) = previous time step
### Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D</td>
<td>= two-dimensional case</td>
</tr>
<tr>
<td>3-D</td>
<td>= three-dimensional case</td>
</tr>
<tr>
<td>P-L</td>
<td>= Power-Law scheme</td>
</tr>
<tr>
<td>SOU</td>
<td>= second-order upwind scheme</td>
</tr>
<tr>
<td>TKE</td>
<td>= Turbulent Kinetic Energy</td>
</tr>
<tr>
<td>SOUM</td>
<td>= P-L plus SOU scheme (SOU-Modified)</td>
</tr>
<tr>
<td>QUICK</td>
<td>= quadratic upstream differencing scheme</td>
</tr>
<tr>
<td>QUICKM</td>
<td>= P-L plus QUICK scheme (QUICK-Modified)</td>
</tr>
</tbody>
</table>
1 INTRODUCTION

1.1 Background

Shelterbelts are used world-wide for such purposes as reduction of soil erosion, control of snow drift, and provision of an effective agrometeorological method for field microclimate management and yield enhancement. The following examples illustrate typical applications of shelterbelts in various disciplines:

1) In agriculture and horticulture, areas where traditional and primitive shelter devices were first invented and applied, the shelterbelt is the most effective tool for providing a protected leeward region for planting crops. There is no doubt that windspeed reduction within a certain area to the leeward side is the primary effect of natural or artificial sheltering objects, although in practice, this major effect will frequently be of less interest than a number of secondary effects such as changes in various factors affecting the growth of plants. Of these factors, temperature, light, water, soil and Carbon dioxide, are all directly influenced by the wind in such a way that reduced windspeed behind the shelterbelt will improve the conditions for plant growth. From a microclimatic point of view, the reduction of heat convection caused by the reduction in wind velocity will cause the temperature of the ground to rise with a consequent increase in temperature of the lowermost layers of air. As a result, during germination and the subsequent period of growth, the plants experience temperatures higher than those generally prevailing in the seasons concerned, resulting in increased crop yields. A reduction in windspeed can also slow down the transmission of water vapor to the upper layers of the air such that
the emission of water to the air, which is tremendously detrimental to the plants, would be decreased. Carbon dioxide, one of the primary growth factors, is strongly affected by wind velocity. A reduction of moderate wind velocities will favor the $CO_2$ conditions of the plant by reducing the waste of $CO_2$ to higher levels of the atmosphere. Furthermore, the light factor is directly influenced by the wind since wind turns the leaves away from their most favorable positions and thereby reduces the quantity of light utilized. Soil drift occurs when the shearing stress on the ground caused by strong winds exceeds soil resistance forces. The detrimental effect on the area from which the soil is drifting will often be so considerable that resowing is required. Repeat soil drifts will cause irredeemable damage to the area robbed of irreplaceable quantities of surface humus. The extent of such damage certainly could be reduced or entirely avoided by a reduction of the wind velocity at the surface of shelter measures [1,2]. From a macroclimatic point of view, strong wind can damage the plants at any growing stage, ranging from complete destruction of germination by simply blowing away the newly sown seeds, to ripping off the flowers and fruits during the ripening period, to causing the lodging of the plant which increases the difficulty of harvest. Strong wind also results in stunted growth of the crop, reducing the size of the fruit and diverting energy into growing the economically useless parts of the plant such as roots and stems instead of growing leaves and fruits. Furthermore, strong wind will repel the pollinating insects and enhance the difficulty of pollination [3].

2) In Geology and Geophysics: Retarding desertification and denudation of the ground resulting from serious soil erosion in tropical desert areas, and field desiccation caused by meager snow accumulation in the continental region are two typical examples of the use of artificial fences and windbreaks [3-6].

3) In environmental control and military applications: Shelterbelts are also a useful method for controlling the spread and concentration of pollutants. In Taiwan, red pine and juniper are planted surrounding the air bases closest to the ocean shore in order to
prevent corrosion and rust on the bodies of military aircraft due to high-density salty see wind [6.7].

4) In civil engineering: a series of snow fences constructed on the sides of highways can efficiently control snow drifting and accumulation, improving the driver’s visibility and decreasing the work involved in maintaining the roads in snowy weather [6. 8-12].

There are many types of shelterbelts in use such as stubble strips, trees or shrubs, fences, reed mats and porous cloth. Many factors affect the choice of a wind break, for instance, establishment and maintenance costs, delay in establishment, portability, shading, water use, and disease and pest control. The value of shelterbelts for agricultural purposes is extremely important, yet many techniques for the design of shelterbelts could hardly be described as quantitative. In fact, in only a few approaches can the claim be made that shelterbelts are sufficiently well-understood to ensure optimum design.

In principle, the aerodynamic influence of the flowfield of a shelterbelt is quite easy to understand. The shelterbelt exerts a force, the so called “drag”, on the flowfield, which is compensated by a loss of fluid momentum. According to Newton’s second law of motion, a reduction in linear momentum of the air when it passes through the shelterbelt implies a deceleration of air particles, and thus the drag is converted into the wind speed reduction desired for sheltering. Obviously, the larger the drag is, the greater is the decrease in wind speed. However, in shelterbelt analysis, researchers are interested in an optimum reduction in a thin air layer near the ground to the lee of the shelterbelt in which crops that need protection are located, rather than the total wind speed reduction in the whole flowfield. Reduction of the wind speed to an expected value at a given point or often a reduction to the highest safe wind speed over the longest possible lee distance, a reduction in evaporation, and an enhancement of CO₂ supply are the typical criteria used in determining the optimum design of shelterbelts. Each of these requirements may have its own most efficient shelter depending upon the different crops planted and various meteorological and soil conditions of the field.
1.2 Literature Review

Basically, there are two main categories of methods for studying shelterbelts: experimental investigation and theoretical calculation. Experimental investigation includes field observation and the wind tunnel test. Theoretical calculations are composed of analytical study and numerical analysis.

1.2.1 Experimental Investigation

As mentioned previously, one of the most important design requirements of the shelterbelt application is how well the leeward windspeed can be reduced below the threshold level over a maximum distance; therefore, discovering a shelterbelt which performs most efficiently yet economically has been the crucial objective of much of the research in shelterbelt analysis for the past half century. Since it is an enormously complex analytical problem which requires a closed system of the full Navier-Stokes equation, most shelterbelt studies have been done, both experimentally and in trial and error, by evaluating the windspeed reduction of existing shelterbelts. Bates (cited by Borrelli et al. [13]) was one of the pioneers in using field experiments to study the effects of height and porosity of shelterbelts on the leeward velocities. Bates observed that an undisturbed wind of 32 kilometers per hour approaching normal to the windbreak of moderate effectiveness can be reduced by the windbreak (in his case, trees) for a distance of 30 heights of the windbreak downwind. Tabler [14] also reported that the drift length on the leeward side of a snow fence, which is defined as either the maximum size of the drift or the distance to the saturation point of snow accumulation, is approximately 30 fence-heights. Similar findings were obtained by Raine and Stevenson [15]. They obtained measurements in a wind tunnel test and found that the area downwind of the model-scale windbreak could be divided into two distinct regions: a triangular region, namely, the quiet zone, whose shape is not a function of the permeability of the shelterbelts and which extends from the
top of the windbreak to the ground at about eight fence-heights behind the windbreak; and a less well-defined region, the wake zone, located above and beyond the quiet zone and stretching downwind to 12 barrier-heights.

Raine and Stevenson also found that inside the quiet zone, the turbulent structure is determined by the bleed flow and thus by fence structures (porosity, height, width, etc.). On the other hand, the longitudinal turbulent fluctuations are more energetic and larger in scale in the wake zone. Perera [16] conducted a wind tunnel test to investigate the shelter effect behind the model-scale fences with different porosities and different shapes of openings and found that it is the porosity instead of the form of the fence which plays the major role in determining the flow structure in the wake zone. McNaughton [17] implemented a series of field experiments to study the effects of windbreaks on turbulent transport and microclimate and confirmed that low turbulent kinetic energy and small eddy size are characteristic of the quiet zone, which is defined as 8-10 shelter-heights in the leeward. Bradley and Mulhearn [19] presented a comprehensive set of wind and shearing stress perturbation data taken downwind of a shelter fence with 50% porosity under the condition of neutral stability in the atmosphere. They showed that the high velocity gradient on the top of the fence caused the wake zone to exhibit large turbulent intensity and to continuously move downstream. Similar comments were made by Finnigan and Bradley [20] who measured the components of the turbulent velocity in the wake of a two-dimensional porous fence under neutral atmospheric conditions in a wind tunnel test. They constructed the downstream budgets of the turbulent kinetic energy by analyzing the observed data within a system of streamline coordinates and stated that pressure transport is the dominant agent in exporting the turbulent energy aloft from regions of strong production in the vicinity of the fence, and the rather prompt increase of TKE at all levels above the fence would be consistent with the pressure transport from a region of enhanced production in the decelerated flow immediately upwind of it. They concluded that the extension of the wake region downstream is due
to the advective term in the TKE (Turbulent Kinetic Energy) budget equation gradually replacing the role of turbulent transports.

In terms of the influence of the leeward windspeed reduction due to the porosity of the shelterbelt, most often quoted are the results of Naegeli who obtained an optimum solution of sorts by showing that a medium dense screen reduces the leeward windspeed at least 20% over a larger distance than either a very dense screen or a screen with high permeability (Eimern et al. [5]). Eimern et al. explained that denser barriers exert greater drag and absorb more momentum from the mean flow, and this momentum deficit, which is clearly indicated by a large velocity reduction either close to the windward or in the near leeward side of the fence, must be balanced by an initial increase in kinetic energy of the fluctuation. As a result, more TKE is produced to alleviate the larger wind shear at the top of the fence, which then leads to a more rapid recovery of the windspeed behind the denser barriers, and thereby a shorter protected area. In contrast to this traditional interpretation, Wilson [21] noted that the observed data both from Raine and Stevenson [15], who conducted a wind tunnel test to study the shelter effects of artificial fences, and Hagen and Skidmore [22], who investigated the windbreak drag influenced by shelter porosity using a field experiment, show a greater wind reduction behind the denser barriers at all distances. Wilson argued that the TKE generated close to the lee side is too small in scale to contribute to the momentum transport, which is believed to cause a faster leeward windspeed recovery, and this induced TKE will dissipate rapidly. In their review document, Heisler and Dewalle [23] attributed this contradiction to failure to observe the similarity requirements between old field experiments and recent research.

Heisler and Dewalle also pointed out that evaluation of windbreak effects should consider the climatology of the approaching wind conditions or at least the wind direction and the atmospheric stability during the periods that the object is to be protected. Wang and Takle [24,25] proclaimed that since wind in the natural environment rarely
blows at a right angle to shelterbelts. Oblique approaching airflow will always redirect itself to minimize the drag force when it penetrates through the barrier. This assertion reconfirmed the statement made by Eimern et al. [5] who explained that shelterbelt modification of the microclimate is highly reliant on the direction of the approaching wind because the drag force exerted by the shelterbelt is strongly dependent upon the wind trajectory through the barrier. Lawrence [26] was one of the first to study changes in shelter effects in oblique wind. During his field experiment, he concluded that the protected region in the lee decreases the more the incoming air stream deviates from the normal direction to the artificial fence. This result was confirmed by Seginer [27] who employed field experiments to study tree windbreaks with 50% porosity instead of wood fences. He reported that windspeed measured along the normal line to the windbreak is strongly dependent on the incident angle of the wind but almost independent of the surface roughness and the shelter effect, which is defined as the lee distance from the barrier (along the center line) in which the windspeed recovers 80% of the undisturbed windspeed at the same height. This protected distance decreases faster than \( \cos \alpha \), where \( \alpha \) is the angle between the approaching wind and the line normal to the shelterbelt.

The longest artificial barrier used in field experiments of oblique angles was probably Jacob’s solid fence with a 2-centimeter thickness, 64-meter width and 2-meter height [28]. Jacob drew the conclusion that at one-third height to the lee, the protected distance decreases much faster than \( \cos(I.A) \), where \( I.A \) is the abbreviation for the incident angle, which is the same as \( \alpha \) defined by Seginer. This observation is consistent with Seginer’s [27], but contradicted Gorsenin’s findings [27]. Nord [29] performed full-scale measurements of wind reduction behind four different types of shelterbelts and reported that when the wind blows at an angle to the shelterbelt, the location of maximum wind reduction in the leeward area moves closer to the belt, the windspeed recovers faster than when it blows at a right angle, and the wind reduction just behind the belt always tends to be greater in oblique than in perpendicular wind.
1.2.2 Theoretical Calculation

Hagen et al. [22] solved the equations of motion using a gradient-diffusion closure scheme where the eddy viscosity was itself modeled (TKE–ε model). A velocity profile, both upstream and in the immediate wake of the fence, was imposed by Hagen et al. to simulate the effect of a porous fence in the atmospheric surface layer. It is well known that the flow pattern is very complex for a barrier with very low porosity. The general remedy for attacking this problem is to subdivide the flow region. For instance, in the analytical theory for the mean velocity behind the obstacle, Counihan et al. [30] split the wake area into a wall zone, a mixing zone and an external zone and claimed that the first order TKE-ε model applied to these three divided flow regions gave satisfactory agreement with the measurements of the flow through 20%, 40% and 60% porous fences. However, it is shown from most field observations that the flow pattern can deviate only very slightly from the well-understood equilibrium surface layer flow and may not cause flow separation for an extremely high porosity barrier such as the practical windbreaks with 20% to 50% porosity.

A detailed examination of the flow through a 50% porous fence carried out by Finnigan and Bradley [20], using as an interpretive framework rigorously derived mean-momentum and turbulent kinetic energy (TKE) equations, indicated a rather complex TKE balance in which the pressure transport (pressure fluctuation - velocity fluctuation correlation) and turbulent transport (velocity triple correlation) played very important roles. Kaiser used an error function to predict the velocity profile by assuming that the shelter effect resulting from the diffusion of the momentum defect is replaced by a passive scalar in the numerical modeling. Although this model is physically unrealistic and somewhat oversimplified, it does reveal that drag force is a decisive factor in shelterbelt analysis [31]. Jackson and Hunt [32] presented an analytical solution for the flow of an adiabatic turbulent boundary on a uniformly rough surface over a two-dimensional hump.
with small curvature. This theory is valid in the limit \( L/y_o \rightarrow \infty \) when \( h/L < \frac{1}{6}(y_o/L)^{0.1} \) and \( \delta/L \gg 2\kappa^2/\ln(\delta/y_o) \) where \( L \) and \( h \) are the characteristic length and height of the hump, respectively. \( y_o \) is the surface roughness length. \( \delta \) is the thickness of the boundary layer and \( \kappa \) is the von Karman constant. They used this analytical approach to show how changes in windspeed and shear stress are related to the size and shape of the hill and to the roughness of the surface. and suggested that this theory may be useful in calculating given roughness estimates of the effect of hills on wind. even though the increase in wind speed over hills is underestimated. Later on, this two-dimensional theory was extended to three-dimensional topography by Mason and Skyes [33].

Wilson and Shaw [34] developed a high-order closure numerical model using the larger plane averaging method to simulate the shelter effect for canopy flow. Unfortunately, although they adjusted the mixing length, their numerical model, consisting of the total turbulence equations, overestimated the turbulence intensity generated by the shelterbelt. Raupach and Shaw [36] used a time/horizontal plane averaging scheme to simulate the sheltering effect. This approach was different from the time/volume averaging technique employed by Finnigan [37]. Li et al. [38] performed a steady state numerical simulation in which the time-averaging followed by volume-averaging method were also used. Both Li et al. [38] and Miller et al. [39] applied a simple small-eddy closure technique (first-order \( K \)-closure) by modifying the mixing length and adding the additional so-called sweep-and-ejection term to obtain better numerical results.

Mellor and Yamada [40] tested a hierarchy of the second-order closure models to parameterize the Reynolds stress terms. Their models are time-dependent, have varying thermal stratification and are able to be extended to the region of the atmosphere where turbulent transfer may be neglected. Yamada [41] utilized a simplified second-moment turbulence closure model to simulate the effects of a tree canopy on air circulation in the atmospheric boundary layer. He stated that qualitative simulation of a canopy flow with nearly constant and low wind speeds in the canopy but large wind shears near the
treetop and unstable or stable temperature layers within the canopy during the night or day are all satisfactory [42]. Wilson [21] used a series of turbulent models ranging from a simple upstream equilibrium-eddy-viscosity scheme ($\mu_t = k_u Z$ model) through a one-equation scheme ($\mu_t - c$ model) and a two-equation scheme (TKE-$c$ model) to second-order accuracy closure schemes in order to compare the patterns of flow through a porous windbreak to those from field experiments. He reported only a slight difference in the prediction of TKE using these numerical models, and the simulations from all turbulence models gave satisfactory agreement with the observed velocity deficits in the near lee wake region of the fence except for underestimations of the speedup region over the fence and the leeward windspeed recovery rate. Attempts to improve these erroneous predictions by including corrections for mean streamline curvature were unsuccessful.

Meyers and Paw [43] used a high-order closure scheme consisting of equations for the mean wind, TKE components and tangential stress, and simplified expressions for the third-order transport terms appearing in the second-order equations in order to simulate the airflow within and above the vegetative environment. They reported that the model in general successfully simulates the speed profiles within and above the vegetative shelters, but the profiles of $\overline{w^2}$ and $\overline{w^2}$ are overestimated near the top of the canopy where both shear and wake production of TKE are high. They believed that these errors are due to incorrect parameterizations for either the dissipation rate of TKE or the pressure-velocity correlations in the budget equations for the second moments. Litvina [44] developed a numerical model of turbulent flow to calculate parameters relating to windspeed reduction, surface shear-stress reduction, and deposition of snow or drifting soil for different species and geometric configurations of sheltering plants. This two-dimensional numerical model solves a system of non-linear equations of velocity components and TKE in the surface layer with closure based on the gradient-diffusion scheme. Alkhalil [4] adopted Litvina's scheme to study the leeward windspeed reduction, extent of protection downwind, TKE distribution and the shelter efficiency related to
the shelter density. Since the original version of Litvina's model performs well for the relatively shorter shelterbelts consisting mainly of annual crops, an assumption of the drag term vanishing right beyond the lee edge of the vegetative barrier is acceptable, but this does not hold true for a taller living-tree shelterbelt with ten-meter height, for example. Therefore, Alkhalil modified this model by extending the influence of the pressure gradient to five shelter-heights downwind in order to remedy the deficiency of the pressure gradient term in the streamwise momentum equation. Wang and Shen [45] developed a simple two-dimensional microcomputer-based shelterbelt model to study the relationships between the permeability of the barriers, atmospheric stability and shelter effects. This model was also used to analyze the effects of shelterbelts on the turbulent exchange coefficient [46], the shelter effects of multiple windbreaks [47] and the protection effects of shelterbelts [48].

The most recent numerical study of shelterbelt analysis was carried out by Wang and Takle [49]. They developed a nonhydrostatic shelterbelt boundary-layer numerical model and used this model to study 1) the characteristics of shelterbelt effects on wind direction and the processes causing changes in wind direction when air passes through and over a porous barrier [25]; 2) the variations of the drag force, permeability and pressure perturbation with width, and the horizontal structure of shelterbelts and their influences on the shelter effects [50]; 3) the relationships between shelter effects and the approach wind incident angle, their spatial variations and their dependence on the density and width of the shelterbelt when the upcoming wind does not blow at a right angle to the barriers [24]; and 4) the patterns and dynamic processes relating to flow interaction with shelterbelts [37].

1.2.3 Summary

Experimental investigation is the most trusted method of obtaining the needed information about a physical process. Data from field observation of shelterbelt flow are
used directly to analyze the shelter effect in all real upstream conditions. However, the drawbacks of field experiments include the difficulties of controlling the specific running conditions and the extremely expensive equipment required for obtaining accurate and extensive information. Useable results cannot be obtained until the experiments have been extended over a number of years sufficient to give a mean value representative of the climate. Furthermore, field data can not be applied to areas with different climate or soil conditions [1].

Running small-scale models in a wind tunnel is another option for studying shelter-belt flows. Although the freestream conditions in the inlet of a wind tunnel are easy to measure and control, the results on small-scale models must be extrapolated to full-scale models following certain similarity rules, which are unavailable in most cases. In addition, due to the complexity of the flow and restrictions in the test section of a wind tunnel, such as the influence of the boundary layer buildup on the tunnel walls and model blockage, the small-scale models are unable to simulate all the features of full-scale shelterbelts. Beyond that, there are many situations where instrumental errors are unavoidable and need to be considered [51].

An analytical study relies on a mathematical model, which consists of a set of differential equations instead of an actual physical model. To avoid unphysical results or over-simplicity, proper assumptions have to be cautiously applied in order to obtain reasonable analytical results (see [30], [32] and [33]). Unfortunately, closed-form solutions can be sought in only a tiny portion of these governing differential equations. Since these solutions often contain very complicated mathematical expressions, their numerical evaluation may present a formidable task. Nevertheless, exact analytical solutions of linear mathematical models are useful for checking the accuracy of other methods [51].

The advantages of a numerical approach include rapid production of results and a relatively low cost compared to the corresponding experimental investigation. as mentioned previously. In a numerical simulation, it is easy to reproduce realistic phenomena
and to provide all the relevant information completely under the preset freestream conditions. However, a perfectly satisfied numerical technique can produce realistic results only if an adequate mathematical model is employed. Sometimes the verification of numerical solutions is very difficult without reference results either from experimental investigation or from some simple theoretical analysis. Thus, after the strengths and weaknesses of both approaches have been studied and analyzed, an optimal prediction effort should include a judicious combination of theoretical analysis and experimental investigation. Following this logic, the current research also progressed through field observation, followed by smoke tunnel flow visualization and numerical analysis. The insight gained in the field observations and the experimental work were important steps to be taken for the successful completion of this project.

1.3 Present Research

The purpose of this study is to formulate an unsteady, three-dimensional, incompressible and control volume approach-based Reynolds averaged Navier-Stokes flow solver to simulate the aerodynamic characteristics of the shelterbelt. Field observations were used to provide the primary air field information of a living-tree shelterbelt under real atmospheric flow conditions, and a wind-tunnel flow visualization of the scale-model fences was conducted to explore the fundamental phenomena of the sheltering flow as well. The Power-Law scheme, which is a popular lower order scheme, neglects turbulence after a certain Peclet number range. This pointed to the need to develop a higher-order numerical scheme that handled turbulent flows well. A modified higher-order numerical scheme using the Lagrange interpolation was therefore developed and applied to simulate the shelterbelt flow field. It is shown that this new scheme, while enhancing the accuracy of computation, also is capable of retaining numerical stability and desirable convergence characteristics which are compromised in most higher-order numerical schemes.
For better prediction, more accurate formulations of the drag terms due to porosity and skin friction, which all artificial windbreaks and living-tree shelterbelts possess, have been included in the momentum equations based on aerodynamic principles. Detailed development of this research is presented in the succeeding chapters. The governing equations and the finite-difference formulation based on the control-volume approach are introduced in Chapter 2. Chapter 3 describes the discretization of the computational domain together with the methodology of discretizing the governing equations. Various turbulent models and the special treatment of the flow field near a solid boundary using wall functions are also introduced: within this context, the development of a modified higher-order accuracy scheme is presented. Details of the field observation of a living-tree shelterbelt in Mead, Nebraska and findings from smoke tunnel flow visualization are reported in Chapter 4. Results from typical two-dimensional and three-dimensional test problems used for code validation are given in Chapter 5. The numerical results from the application of this method to shelterbelt aerodynamic analysis is presented in Chapter 6. Finally, the conclusions of this research and recommendations for further work are discussed in Chapter 7.
2 THEORETICAL BACKGROUND

2.1 Governing Equations

The governing differential equations of fluid flow based on the physical principles of conservation can be cast into the general form:

\[ \frac{\partial}{\partial t}(\rho \Phi) + \nabla \cdot \vec{J} = S_\Phi \]  \hspace{1cm} (2.1)

where the total flux \( \vec{J} \), the convective flux \( \vec{F}_c \), and the diffusive flux \( \vec{F}_d \) are defined as:

\[ \vec{J} = \vec{F}_c + \vec{F}_d \]  \hspace{1cm} (2.2)

\[ \vec{F}_c = \rho \vec{V} \Phi \]  \hspace{1cm} (2.3)

\[ \vec{F}_d = -\Gamma \nabla \Phi \]  \hspace{1cm} (2.4)

Substituting Equations 2.2, 2.3, and 2.4 into Equation 2.1 yields:

\[ \frac{\partial}{\partial t}(\rho \Phi) + \nabla \cdot (\rho \vec{V} \Phi - \Gamma \nabla \Phi) = S_\Phi \]  \hspace{1cm} (2.5)

Equation 2.1 is a general differential equation governing convection-diffusion processes. The Reynolds averaged three-dimensional unsteady incompressible form of the differential equations governing the conservation of mass and the conservation of momentum in a Cartesian coordinate takes the form:

mass equation:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \]  \hspace{1cm} (2.6)
u-momentum equation:

\[
\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u u)}{\partial x} + \frac{\partial (\rho u v)}{\partial y} + \frac{\partial (\rho u w)}{\partial z} = \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial u}{\partial z})
- \frac{\partial p}{\partial x} + \left[ \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial z}) \right]
\]  

(2.7)

v-momentum equation:

\[
\frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho v u)}{\partial x} + \frac{\partial (\rho v v)}{\partial y} + \frac{\partial (\rho v w)}{\partial z} = \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial v}{\partial z})
- \frac{\partial p}{\partial y} + \left[ \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial z}) \right]
\]  

(2.8)

w-momentum equation:

\[
\frac{\partial (\rho w)}{\partial t} + \frac{\partial (\rho w u)}{\partial x} + \frac{\partial (\rho w v)}{\partial y} + \frac{\partial (\rho w w)}{\partial z} = \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial w}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial w}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial z})
- \frac{\partial p}{\partial z} + \left[ \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial z}) \right]
\]  

(2.9)

where the effective viscosity, \( \mu_{\text{eff}} \), is the sum of the laminar molecular viscosity, \( \mu_l \), and the turbulent dynamic viscosity, \( \mu_t \), i.e.,

\[
\mu_{\text{eff}} = \mu_l + \mu_t
\]  

(2.10)

2.2 Control Volume Approach

2.2.1 Integral Momentum Equations

By grouping the convective flux and the diffusive flux together, the u-momentum equation can be rearranged as:

\[
\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u u - \mu_{\text{eff}} \frac{\partial u}{\partial x})}{\partial x} + \frac{\partial (\rho u v - \mu_{\text{eff}} \frac{\partial u}{\partial y})}{\partial y} + \frac{\partial (\rho u w - \mu_{\text{eff}} \frac{\partial u}{\partial z})}{\partial z} = \frac{\partial p}{\partial x}
+ \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial z})
\]  

(2.11)
Integrating over the control surfaces of the typical u-momentum control volume, the u-momentum equation becomes:

\[
\begin{align*}
\int_{t_0}^t \int_s^e \int_w^f \int_b^a \left( \frac{\partial (\rho u)}{\partial t} + \frac{\partial}{\partial x} (\rho u u - \mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\rho u v - \mu_{eff} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial z} (\rho u w - \mu_{eff} \frac{\partial u}{\partial z}) \right) dx dy dz dt
\end{align*}
\]

\[
= \int_{t_0}^t \int_s^e \int_w^f (- \frac{\partial p}{\partial x}) dx dy dz dt
\]

\[
+ \int_{t_0}^t \int_s^e \int_w^f \left[ \frac{\partial}{\partial x} (\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{eff} \frac{\partial w}{\partial z}) \right] dx dy dz dt
\]  

(2.12)

Therefore, the integral formulation of the u-momentum equation is given as:

\[
\{(\rho u) - (\rho u)^0\} \left( \frac{\Delta x \Delta y \Delta z}{\Delta t} \right) + \alpha (J_{ue} - J_{uw} + J_{un} - J_{us} + J_{ut} - J_{ub})
\]

\[
+ (1 - \alpha) (J_{ue}^0 - J_{uw}^0 + J_{un}^0 - J_{us}^0 + J_{ut}^0 - J_{ub}^0)
\]

\[
= (p_w - p_e) \Delta y \Delta z + S_u (\Delta x \Delta y \Delta z)
\]  

(2.13)

where \( S_u (\Delta x \Delta y \Delta z) \) represents the integral source term due to the turbulent flow and \( S_u \) is the mean value of the turbulence source in the u-momentum control volume. Following the same procedures, the v-momentum can be rewritten as:

\[
\frac{\partial (\rho v)}{\partial t} + \frac{\partial}{\partial x} (\rho vu - \mu_{eff} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y} (\rho vv - \mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\rho vw - \mu_{eff} \frac{\partial v}{\partial z}) = - \frac{\partial p}{\partial y}
\]

\[
+ \frac{\partial}{\partial x} (\mu_{eff} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial y} (\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{eff} \frac{\partial w}{\partial y})
\]  

(2.14)

By integrating the above equation over the control surfaces of the typical v-momentum control volume, the v-momentum equation becomes:

\[
\begin{align*}
\int_{t_0}^t \int_s^e \int_w^f \int_b^a \left( \frac{\partial (\rho v)}{\partial t} + \frac{\partial}{\partial x} (\rho vu - \mu_{eff} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y} (\rho vv - \mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\rho vw - \mu_{eff} \frac{\partial v}{\partial z}) \right) dx dy dz dt
\end{align*}
\]

\[
= \int_{t_0}^t \int_s^e \int_w^f (- \frac{\partial p}{\partial y}) dx dy dz dt
\]

\[
+ \int_{t_0}^t \int_s^e \int_w^f \left[ \frac{\partial}{\partial x} (\mu_{eff} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial y} (\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{eff} \frac{\partial w}{\partial y}) \right] dx dy dz dt
\]  

(2.15)

Therefore, the integral formulation of the v-momentum equation is shown as:

\[
\{(\rho v) - (\rho v)^0\} \left( \frac{\Delta x \Delta y \Delta z}{\Delta t} \right) + \alpha (J_{ve} - J_{vw} + J_{vn} - J_{vs} + J_{vt} - J_{vb})
\]
\[(1 - \alpha)(J_{x0} - J_{w0} + J_{x0} - J_{x0} + J_{v0} - J_{y0}) = (p_s - p_n)(\Delta x \Delta z) + S_v(\Delta x \Delta y \Delta z) \quad (2.16)\]

Similarly, the conservative form of the w-momentum equation based on a Cartesian coordinate system is given by:

\[
\frac{\partial (\rho w)}{\partial t} + \frac{\partial (\rho w u)}{\partial x} + \frac{\partial (\rho w v)}{\partial y} + \frac{\partial (\rho w w)}{\partial z} - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial w}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial w}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial z}) = 0 \quad (2.17)
\]

After being integrated over the control surfaces of the typical w-momentum control volume, the w-momentum equation becomes:

\[
\int_{t_0}^{t} \int_{s}^{s} \int_{w}^{w} \left[ \frac{\partial (\rho w u)}{\partial t} + \frac{\partial (\rho w u)}{\partial x} + \frac{\partial (\rho w v)}{\partial y} + \frac{\partial (\rho w w)}{\partial z} - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial w}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial w}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial z}) \right] dxdydzdt
\]

\[
= \int_{t_0}^{t} \int_{s}^{s} \int_{w}^{w} \left[ - \frac{\partial p}{\partial z} \right] dxdydzdt
\]

\[
+ \int_{t_0}^{t} \int_{s}^{s} \int_{w}^{w} \left[ \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial z}) \right] dxdydzdt \quad (2.18)
\]

Hence, the final expression of the integral w-momentum equation is given by:

\[
\{(\rho w) - (\rho w^0)\} \left( \frac{\Delta x \Delta y \Delta z}{\Delta t} \right) + \alpha (J_{w0} - J_{w0} + J_{x0} - J_{x0} + J_{w0} - J_{w0})
\]

\[
+ (1 - \alpha)(J_{w0} - J_{w0} + J_{x0} - J_{x0} + J_{w0} - J_{w0}) = (p_s - p_n)(\Delta x \Delta y \Delta z) \quad (2.19)
\]

where the superscript 0 stands for the previous time step value, and \(\alpha\) is a factor which represents explicit, implicit, or Crank-Nicolson schemes depending on whether \(\alpha\) equals 0, 1, or 0.5, respectively. Fully implicit formulations are used for representing the pressure gradients and the source terms due to the turbulent flow. \(J\)'s are the momentum fluxes defined as:

\[
J_{w0} = [(\rho u u)_w - (\mu_{eff} \frac{\partial u}{\partial x})_w](\Delta y \Delta z) : \quad J_{w0} = [(\rho u u)_w - (\mu_{eff} \frac{\partial u}{\partial x})_w](\Delta y \Delta z)
\]
\[ J_{un} = [(\rho u)v]_n - (\mu eff \frac{\partial u}{\partial y})_n(\Delta x \Delta z) : \quad J_{us} = [(\rho u)v]_s - (\mu eff \frac{\partial u}{\partial y})_s(\Delta x \Delta z) \]
\[ J_{ut} = [(\rho u)w]_t - (\mu eff \frac{\partial u}{\partial z})_t(\Delta x \Delta y) : \quad J_{ub} = [(\rho u)w]_b - (\mu eff \frac{\partial u}{\partial z})_b(\Delta x \Delta y) \]
\[ J_{ve} = [(\rho v)u]_e - (\mu eff \frac{\partial v}{\partial x})_e(\Delta y \Delta z) : \quad J_{vw} = [(\rho v)u]_w - (\mu eff \frac{\partial v}{\partial x})_w(\Delta y \Delta z) \]
\[ J_{vn} = [(\rho v)n]_n - (\mu eff \frac{\partial v}{\partial y})_n(\Delta x \Delta z) : \quad J_{vs} = [(\rho v)n]_s - (\mu eff \frac{\partial v}{\partial y})_s(\Delta x \Delta z) \]
\[ J_{vt} = [(\rho v)t]_t - (\mu eff \frac{\partial v}{\partial z})_t(\Delta x \Delta y) : \quad J_{vb} = [(\rho v)t]_b - (\mu eff \frac{\partial v}{\partial z})_b(\Delta x \Delta y) \]
\[ J_{we} = [(\rho w)u]_e - (\mu eff \frac{\partial w}{\partial x})_e(\Delta y \Delta z) : \quad J_{ww} = [(\rho w)u]_w - (\mu eff \frac{\partial w}{\partial x})_w(\Delta y \Delta z) \]
\[ J_{wn} = [(\rho w)n]_n - (\mu eff \frac{\partial w}{\partial y})_n(\Delta x \Delta z) : \quad J_{ws} = [(\rho w)n]_s - (\mu eff \frac{\partial w}{\partial y})_s(\Delta x \Delta z) \]
\[ J_{wt} = [(\rho w)t]_t - (\mu eff \frac{\partial w}{\partial z})_t(\Delta x \Delta y) : \quad J_{wb} = [(\rho w)t]_b - (\mu eff \frac{\partial w}{\partial z})_b(\Delta x \Delta y) \]

### 2.2.2 Integral Mass Equation

The continuity equation is rewritten here as follows:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \quad (2.20) \]

Again, integrating the continuity equation with respect to the typical main control volume, the integral form of the mass equation is given as:

\[ \int_{t_0}^{t_1} \int_{s_0}^{s_1} \int_{w_0}^{w_1} \int_{b_0}^{b_1} \left( \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} \right) dx dy dz dt = 0 \quad (2.21) \]

or

\[ (\rho - \rho^0)(\frac{\Delta x \Delta y \Delta z}{\Delta t}) + \alpha(F_e - F_w + F_n - F_s + F_t - F_b) \]
\[ + (1 - \alpha)(F^0_e - F^0_w + F^0_n - F^0_s + F^0_t - F^0_b) = 0 \quad (2.22) \]

where \( F \)'s are the mass flow rates through the control surfaces defined as:

\[ F_e = (\rho u)e(\Delta y \Delta z) : \quad F_w = (\rho u)w(\Delta y \Delta z) \]
\[ F_n = (\rho v)n(\Delta x \Delta z) : \quad F_s = (\rho v)s(\Delta x \Delta z) \]
\[
F_i = (\rho w)_i(\Delta x \Delta y) : \quad F_b = (\rho w)_b(\Delta x \Delta y)
\]

and
\[
F^o_e = (\rho u)^o_e(\Delta y \Delta z) ; \quad F^o_w = (\rho u)^o_w(\Delta y \Delta z) \\
F^o_n = (\rho v)^o_n(\Delta x \Delta z) ; \quad F^o_s = (\rho v)^o_s(\Delta x \Delta z) \\
F^o_i = (\rho w)^o_i(\Delta x \Delta y) ; \quad F^o_b = (\rho w)^o_b(\Delta x \Delta y)
\]
3 NUMERICAL ALGORITHM

The numerical algorithm, based on Patankar’s SIMPLER [51], uses a staggered grid arrangement and a brief explanation for the choice of grid system follows.

3.1 Domain Discretization

A non-staggered arrangement which stores both pressure and velocity components at the same grid (primary grid) has the following drawbacks: first, the momentum equation or continuity equation will contain the pressure difference or velocity gradient between two alternate grids instead of two adjacent ones. This situation reduces the accuracy of the solution, since a coarse grid arrangement is used. Secondly, it may produce a zig-zag pressure field or a wavy velocity field for one dimensional flow or a checkerboard pressure pattern or velocity field for two dimensional flow. One way to eliminate these unrealistic results is to store velocity components at locations different from those of pressure. The locations of velocity components are staggered in the coordinate direction and stored on the surface of the control volume (or on the auxiliary grid). Therefore, the velocities can be directly used for continuity calculation without using interpolation, and the pressure gradient can be evaluated between two neighboring grid points.

The grid configuration used in this research is shown in Figure 3.1. In this arrangement, pressure, TKE, and $\epsilon$ are stored at the primary grids, and the velocity components are stored at the auxiliary grids on the faces of the control volume.
Figure 3.1 Staggered grid configuration
3.2 Discretization Methodology

3.2.1 Integral Governing Equations

The integral governing equations derived in Chapter 2 are summarized here:

mass equation:

$$\{(\rho - \rho^0)\left(\frac{\Delta x \Delta y \Delta z}{\Delta t}\right) + \alpha (F_e - F_w + F_n - F_s + F_t - F_b)\}$$

$$+ (1 - \alpha)(F_e^0 - F_w^0 + F_n^0 - F_s^0 + F_t^0 - F_b^0) = 0$$

(3.1)

For incompressible flow, mass equation becomes:

$$F_e - F_w + F_n - F_s + F_t - F_b = 0$$

(3.2)

or

$$F_e^0 - F_w^0 + F_n^0 - F_s^0 + F_t^0 - F_b^0 = 0$$

(3.3)

u-momentum equation:

$$\{(\rho u) - (\rho u)^0\left(\frac{\Delta x \Delta y \Delta z}{\Delta t}\right) + \alpha (J_{ue} - J_{uw} + J_{un} - J_{us} + J_{ut} - J_{ub})\}$$

$$+ (1 - \alpha)(J_{ue}^0 - J_{uw}^0 + J_{un}^0 - J_{us}^0 + J_{ut}^0 - J_{ub}^0)$$

$$= (p_w - p_e)\Delta y \Delta z + S_u(\Delta x \Delta y \Delta z)$$

(3.4)

v-momentum equation:

$$\{(\rho v) - (\rho v)^0\left(\frac{\Delta x \Delta y \Delta z}{\Delta t}\right) + \alpha (J_{ve} - J_{vw} + J_{vn} - J_{vs} + J_{vt} - J_{vb})\}$$

$$+ (1 - \alpha)(J_{ve}^0 - J_{vw}^0 + J_{vn}^0 - J_{vs}^0 + J_{vt}^0 - J_{vb}^0)$$

$$= (p_s - p_n)\Delta x \Delta z + S_v(\Delta x \Delta y \Delta z)$$

(3.5)

w-momentum equation:

$$\{(\rho w) - (\rho w)^0\left(\frac{\Delta x \Delta y \Delta z}{\Delta t}\right) + \alpha (J_{we} - J_{ww} + J_{wn} - J_{ws} + J_{wt} - J_{wb})\}$$

$$+ (1 - \alpha)(J_{we}^0 - J_{ww}^0 + J_{wn}^0 - J_{ws}^0 + J_{wt}^0 - J_{wb}^0)$$

$$= (p_b - p_l)\Delta x \Delta y + S_w(\Delta x \Delta y \Delta z)$$

(3.6)
3.2.2 Discretization of Momentum Equations

Subtracting Equation 3.4 from both Equation 3.2 multiplied by \((\alpha \cdot u_P)\) and Equation 3.3 multiplied by \((1 - \alpha) \cdot u_P^0\), the u-momentum equation becomes:

\[
(u - u^0)\rho \Delta x \Delta y \Delta z + \alpha (J_{ue} - F_{e}u_P) - (J_{uw} - F_{w}u_P) \\
+ (J_{un} - F_{n}u_P) - (J_{us} - F_{s}u_P) + (J_{ut} - F_{t}u_P) - (J_{ub} - F_{b}u_P) \\
+ (1 - \alpha) [(J_{ue}^0 - F_{e}^0u_P^0) - (J_{uw}^0 - F_{w}^0u_P^0) \\
+ (J_{un}^0 - F_{n}^0u_P^0) - (J_{us}^0 - F_{s}^0u_P^0) + (J_{ut}^0 - F_{t}^0u_P^0) - (J_{ub}^0 - F_{b}^0u_P^0)] \\
= (p_w - p_e) \Delta y + S_u \Delta x \Delta y \Delta z \\
(3.7)
\]

Following the same manipulation described by Patankar [51], we can express the total flux across the interface in terms of the main variables stored on the two adjacent grid points. Therefore the total flux on each control surface based on the seven-grid stencil shown in Figure 3.2 can be written as:

\[
J_{ue} - F_{e}u_P = a_{uE}(u_P - u_E) \\
J_{un} - F_{n}u_P = a_{uN}(u_P - u_N) \\
J_{ut} - F_{t}u_P = a_{uT}(u_P - u_T) \\
J_{ub} - F_{b}u_P = a_{uB}(u_P - u_B)
\]

and

\[
J_{ue}^0 - F_{e}^0u_P^0 = a_{uE}^0(u_P^0 - u_E^0) \\
J_{un}^0 - F_{n}^0u_P^0 = a_{uN}^0(u_P^0 - u_N^0) \\
J_{ut}^0 - F_{t}^0u_P^0 = a_{uT}^0(u_P^0 - u_T^0) \\
J_{ub}^0 - F_{b}^0u_P^0 = a_{uB}^0(u_P^0 - u_B^0)
\]

Substituting all above expressions into Equation (3.7) and rearranging, the discretized u-momentum equation becomes:

\[
\tilde{a}_{uP} u_P = \sum (\tilde{a}_{unb} u_{nb}) + b_u + (p_w - p_e)(\Delta y \Delta z) + S_u(\Delta x \Delta y \Delta z) \\
(3.8)
\]
Figure 3.2 Typical three-dimensional stencil
where $\tilde{a}_{up}$ is the coefficient at the grid point $P$, and $\tilde{a}_{uE}$, $\tilde{a}_{uW}$, $\tilde{a}_{uN}$, $\tilde{a}_{uS}$, $\tilde{a}_{uT}$, $\tilde{a}_{uB}$ are the neighboring coefficients given by:

\[
\begin{align*}
\tilde{a}_{uE} &= \alpha \cdot a_{uE} : \quad \tilde{a}_{uW} = \alpha \cdot a_{uW} \\
\tilde{a}_{uN} &= \alpha \cdot a_{uN} : \quad \tilde{a}_{uS} = \alpha \cdot a_{uS} \\
\tilde{a}_{uT} &= \alpha \cdot a_{uT} : \quad \tilde{a}_{uB} = \alpha \cdot a_{uB} \\
\tilde{a}_{up} &= \frac{\rho \Delta x \Delta y \Delta z}{\Delta t} + \alpha(a_{uE} + a_{uW} + a_{uN} + a_{uS} + a_{uT} + a_{uB}) \\
b_u &= (1 - \alpha)[a_{uE}^0 u_E^0 + a_{uW}^0 u_W^0 + a_{uN}^0 u_N^0 + a_{uS}^0 u_S^0 + a_{uT}^0 u_T^0 + a_{uB}^0 u_B^0] \\
& \quad - (a_{uE}^0 + a_{uW}^0 + a_{uN}^0 + a_{uS}^0 + a_{uT}^0 + a_{uB}^0) u_p^0 \\
& \quad + \left(\frac{\rho \Delta x \Delta y \Delta z}{\Delta t}\right) u_p^0 + (p_w - p_e) \Delta x \Delta y \Delta z + S_u \Delta x \Delta y \Delta z
\end{align*}
\]

and

\[
\begin{align*}
a_{uE} &= D_{ue} A(|P_{ue}|) + [-F_{ue} \cdot 0] : \quad a_{uW} = D_{uw} A(|P_{uw}|) + [F_{uw} \cdot 0] \\
a_{uN} &= D_{un} A(|P_{un}|) + [-F_{un} \cdot 0] : \quad a_{uS} = D_{us} A(|P_{us}|) + [F_{us} \cdot 0] \\
a_{uT} &= D_{ut} A(|P_{ut}|) + [-F_{ut} \cdot 0] : \quad a_{uB} = D_{ub} A(|P_{ub}|) + [F_{ub} \cdot 0] \\
a_{uE}^0 &= D_{ue}^0 A(|P_{ue}^0|) + [-F_{ue}^0 \cdot 0] : \quad a_{uW}^0 = D_{uw}^0 A(|P_{uw}^0|) + [F_{uw}^0 \cdot 0] \\
a_{uN}^0 &= D_{un}^0 A(|P_{un}^0|) + [-F_{un}^0 \cdot 0] : \quad a_{uS}^0 = D_{us}^0 A(|P_{us}^0|) + [F_{us}^0 \cdot 0] \\
a_{uT}^0 &= D_{ut}^0 A(|P_{ut}^0|) + [-F_{ut}^0 \cdot 0] : \quad a_{uB}^0 = D_{ub}^0 A(|P_{ub}^0|) + [F_{ub}^0 \cdot 0] \\
\end{align*}
\]

where the flow rate $F$'s through the control volume faces are defined as shown in section 2.2.2. and $D$ the diffusion conductances on control volume faces are given as follows:

\[
\begin{align*}
D_{ue} &= (\mu_{eff} \frac{\Delta y \Delta z}{\Delta x})_e : \quad D_{uw} = (\mu_{eff} \frac{\Delta y \Delta z}{\Delta x})_w \\
D_{un} &= (\mu_{eff} \frac{\Delta x \Delta z}{\Delta y})_n : \quad D_{us} = (\mu_{eff} \frac{\Delta x \Delta z}{\Delta y})_s \\
D_{ut} &= (\mu_{eff} \frac{\Delta x \Delta y}{\Delta z})_t : \quad D_{ub} = (\mu_{eff} \frac{\Delta x \Delta y}{\Delta z})_b
\end{align*}
\]
The Peclet numbers on control surfaces are given by:

\[ P = \frac{F}{D} : \quad P^0 = \frac{F^0}{D^0} \]

and \( A(|P|) \) is a function based on the Power-Law scheme given by:

\[ A(|P|) = \left[ 0.1 - 0.1 |P|^5 \right] : \quad A(|P^0|) = \left[ 0.1 - 0.1 |P^0|^5 \right] \]

where \([ \ldots \] indicates the greater of the quantities within.

Similarly, the discretized v-momentum and w-momentum equations are given as:

\[ \bar{a}_{vP} v_P = \sum (\bar{a}_{vnb} v_{nb}) + b_v + (p_t - p_n)(\Delta x \Delta y \Delta z) + S_v(\Delta x \Delta y \Delta z) \quad (3.9) \]

and

\[ \bar{a}_{wP} w_P = \sum (\bar{a}_{wnb} w_{nb}) + b_w + (p_t - p_h)(\Delta x \Delta y \Delta z) + S_w(\Delta x \Delta y \Delta z) \quad (3.10) \]

The coefficients are similar to those of the u-momentum equation based on each equation's specific control volume.

### 3.2.3 Discretization of the Pressure Equation

The integration of mass conservation with respect to a typical control volume yields:

\[ (\rho u)_{x \Delta y \Delta z} - (\rho u)_{w \Delta y \Delta z} + (\rho v)_{n \Delta x \Delta z} \]

\[ -(\rho v)_{z \Delta x \Delta z} + (\rho w)_{i \Delta x \Delta y} - (\rho w)_{b \Delta x \Delta y} = 0 \quad (3.11) \]

From Equations 3.8, 3.9 and 3.10, the momentum equations written for the u, v, and w control volumes associated with the main control volume become:

\[ \bar{a}_{ue} u_e = \left( \sum \bar{a}_{unb} u_{nb} \right)_e - (p_E - p_P) \Delta y \Delta z + S_{ue}(\Delta x \Delta y \Delta z) \quad (3.12) \]
\[
\ddot{a}_{uw}u_w = (\sum \ddot{a}_{unb}u_{nb})_w - (p_p - p_w)\Delta y\Delta z + S_{uw}(\Delta x\Delta y\Delta z) \quad (3.13)
\]

\[
\ddot{a}_{vn}v_n = (\sum \ddot{a}_{vn}v_{nb})_n - (p_n - p_p)\Delta x\Delta z + S_{vn}(\Delta x\Delta y\Delta z) \quad (3.14)
\]

\[
\ddot{a}_{vs}v_s = (\sum \ddot{a}_{vs}v_{nb})_s - (p_p - p_s)\Delta x\Delta z + S_{vs}(\Delta x\Delta y\Delta z) \quad (3.15)
\]

\[
\ddot{a}_{wt}w_t = (\sum \ddot{a}_{wt}w_{nb})_t - (p_T - p_p)\Delta x\Delta y + S_{wt}(\Delta x\Delta y\Delta z) \quad (3.16)
\]

\[
\ddot{a}_{wb}w_b = (\sum \ddot{a}_{wb}w_{nb})_b - (p_p - p_B)\Delta x\Delta y + S_{wb}(\Delta x\Delta y\Delta z) \quad (3.17)
\]

Equations 3.12 to 3.17 can be rewritten as:

\[
u_e = \dot{u}_e - (p_E - p_P)\Delta y\Delta z / \ddot{a}_{ue} \quad (3.18)
\]

\[
u_w = \dot{u}_w - (p_p - p_w)\Delta y\Delta z / \ddot{a}_{uw} \quad (3.19)
\]

\[
v_n = \dot{v}_n - (p_n - p_p)\Delta x\Delta z / \ddot{a}_{vn} \quad (3.20)
\]

\[
v_s = \dot{v}_s - (p_p - p_s)\Delta x\Delta z / \ddot{a}_{vs} \quad (3.21)
\]

\[
w_t = \dot{w}_t - (p_T - p_p)\Delta x\Delta y / \ddot{a}_{wt} \quad (3.22)
\]

\[
w_b = \dot{w}_b - (p_p - p_B)\Delta x\Delta y / \ddot{a}_{wb} \quad (3.23)
\]

where the pseudo-velocities on the control surfaces are defined by:

\[
\dot{u}_e = \frac{1}{\ddot{a}_{ue}}(\sum \ddot{a}_{unb}u_{nb})_e + S_{ue}(\Delta x\Delta y\Delta z) / \ddot{a}_{ue} \quad (3.24)
\]

\[
\dot{u}_w = \frac{1}{\ddot{a}_{uw}}(\sum \ddot{a}_{unb}u_{nb})_w + S_{uw}(\Delta x\Delta y\Delta z) / \ddot{a}_{uw} \quad (3.25)
\]

\[
\dot{v}_n = \frac{1}{\ddot{a}_{vn}}(\sum \ddot{a}_{vn}v_{nb})_n + S_{vn}(\Delta x\Delta y\Delta z) / \ddot{a}_{vn} \quad (3.26)
\]

\[
\dot{v}_s = \frac{1}{\ddot{a}_{vs}}(\sum \ddot{a}_{vs}v_{nb})_s + S_{vs}(\Delta x\Delta y\Delta z) / \ddot{a}_{vs} \quad (3.27)
\]

\[
\dot{w}_t = \frac{1}{\ddot{a}_{wt}}(\sum \ddot{a}_{wt}w_{nb})_t + S_{wt}(\Delta x\Delta y\Delta z) / \ddot{a}_{wt} \quad (3.28)
\]

\[
\dot{w}_b = \frac{1}{\ddot{a}_{wb}}(\sum \ddot{a}_{wb}w_{nb})_b + S_{wb}(\Delta x\Delta y\Delta z) / \ddot{a}_{wb} \quad (3.29)
\]
Substituting Equations 3.18 through 3.23 into Equation 3.11, the pressure equation becomes:

\[ a_{pP} \rho p \rho p = a_{pE} \rho E + a_{pW} \rho W + a_{pN} \rho N + a_{pS} \rho S + a_{pT} \rho T + a_{pB} \rho B + b_p \]

\[ \Rightarrow a_{pP} \rho p = \sum (a_{pnb} \rho nb) + b_p \] (3.30)

where the coefficients are defined by:

\[ a_{pE} = \frac{\rho_e}{\overline{\alpha_{ue}}} (\Delta y \Delta z)^2 \quad a_{pW} = \frac{\rho_w}{\overline{\alpha_{uw}}} (\Delta y \Delta z)^2 \]
\[ a_{pN} = \frac{\rho_n}{\overline{\alpha_{vn}}} (\Delta x \Delta z)^2 \quad a_{pS} = \frac{\rho_s}{\overline{\alpha_{vs}}} (\Delta x \Delta z)^2 \]
\[ a_{pT} = \frac{\rho_t}{\overline{\alpha_{wt}}} (\Delta x \Delta y)^2 \quad a_{pB} = \frac{\rho_b}{\overline{\alpha_{ub}}} (\Delta x \Delta y)^2 \]

\[ a_{pP} = \sum a_{pnb} = a_{pE} + a_{pW} + a_{pN} + a_{pS} + a_{pT} + a_{pB} \]

Therefore the final source term \( b_p \) in the pressure equation is given by:

\[ b_p = -\hat{u}_x \Delta y \Delta z + \hat{u}_w \Delta y \Delta z - \hat{v}_n \Delta x \Delta z + \hat{v}_s \Delta x \Delta z - \hat{w}_t \Delta x \Delta y + \hat{w}_b \Delta x \Delta y \] (3.31)

### 3.2.4 Discretization of the Pressure Correction Equation

The momentum equations can be solved only when the pressure field is given or somehow estimated. Unless the correct pressure field is employed, the resulting velocity field will not satisfy the continuity equation. The velocity field based on a guessed (or calculated) pressure field \( \rho^* \) will be denoted by \( u^*, v^* \) and \( w^* \). These “starred” velocity fields can be obtained by solving the discretized momentum equations given below:

\[ \tilde{a}_{uP} \hat{u}_p = \sum \tilde{a}_{unb} \hat{u}_{nb}^* - (p_e^* - p_w^*) \Delta y \Delta z + S_{u}^* (\Delta x \Delta y \Delta z) \] (3.32)
\[ \tilde{a}_{vP} \hat{v}_p = \sum \tilde{a}_{vnb} \hat{v}_{nb}^* - (p_n^* - p_s^*) \Delta x \Delta z + S_{v}^* (\Delta x \Delta y \Delta z) \] (3.33)
\[ \tilde{a}_{wP} \hat{w}_p = \sum \tilde{a}_{wnb} \hat{w}_{nb}^* - (p_t^* - p_b^*) \Delta x \Delta y + S_{w}^* (\Delta x \Delta y \Delta z) \] (3.34)

Using the procedures adopted from Patanker [51], the imperfect * quantities are corrected in the following manner:
where $u'$, $v'$ and $w'$ are the pressure and velocity correction terms, respectively.

Subtracting Equation 3.8 from Equation 3.32, Equation 3.9 from Equation 3.33 and Equation 3.10 from Equation 3.34 yields:

\[
\begin{align*}
\bar{a}_{uP} \quad u'_{P} &= \sum \bar{a}_{u_{nb}}u'_{n_{b}} - (p'_e - p'_w)\Delta y\Delta z + S'_u(\Delta x\Delta y\Delta z) \\
\bar{a}_{vP} \quad v'_{P} &= \sum \bar{a}_{v_{nb}}v'_{n_{b}} - (p'_n - p'_s)\Delta x\Delta z + S'_v(\Delta x\Delta y\Delta z) \\
\bar{a}_{wP} \quad w'_{P} &= \sum \bar{a}_{w_{nb}}w'_{n_{b}} - (p'_i - p'_b)\Delta x\Delta y + S'_w(\Delta x\Delta y\Delta z)
\end{align*}
\]

Neglecting the first term and the third term on the right hand side in Equations 3.38 to 3.40, which represent the implicit influence of the pressure corrections on the velocity fields, the velocity correction terms become:

\[
\begin{align*}
u'_{P} &= -(p'_e - p'_w)\Delta y\Delta z / \bar{a}_{uP} \\
v'_{P} &= -(p'_n - p'_s)\Delta x\Delta z / \bar{a}_{vP} \\
w'_{P} &= -(p'_i - p'_b)\Delta x\Delta y / \bar{a}_{wP}
\end{align*}
\]

Substituting Equation 3.41 into Equation 3.35, Equation 3.42 into Equation 3.36 and Equation 3.43 into Equation 3.37, the corrected velocities are obtained as:

\[
\begin{align*}
u_{P} &= u'_{P} - (p'_e - p'_w)\Delta y\Delta z / \bar{a}_{uP} \\
v_{P} &= v'_{P} - (p'_n - p'_s)\Delta x\Delta z / \bar{a}_{vP} \\
w_{P} &= w'_{P} - (p'_i - p'_b)\Delta x\Delta y / \bar{a}_{wP}
\end{align*}
\]
Also, the corrected velocity on each control surface can be expressed as:

\[ u_e = u_e^* - (p'_E - p'_P)\Delta y\Delta z / \bar{a}_{ue} \]  (3.47)

\[ u_w = u_w^* - (p'_P - p'_W)\Delta y\Delta z / \bar{a}_{uw} \]  (3.48)

\[ u_n = u_n^* - (p'_N - p'_P)\Delta x\Delta z / \bar{a}_{un} \]  (3.49)

\[ u_s = u_s^* - (p'_P - p'_S)\Delta x\Delta z / \bar{a}_{us} \]  (3.50)

\[ w_t = w_t^* - (p'_T - p'_P)\Delta x\Delta y / \bar{a}_{wt} \]  (3.51)

\[ w_b = w_b^* - (p'_P - p'_B)\Delta x\Delta y / \bar{a}_{wb} \]  (3.52)

Following the same procedure adopted for the derivation of the pressure equation by substituting Equation 3.47 through Equation 3.52 into the continuity equation. Equation 3.11, the pressure correction equation is obtained as:

\[ a_{pP} p'_P = a_{pE} p'_E + a_{pW} p'_W + a_{pN} p'_N + a_{pS} p'_S + a_{pT} p'_T + a_{pB} p'_B + b_{p'} \]

\[ \Rightarrow \quad a_{pP} p'_P = \sum (a_{pm} p'_m) + b_{p'} \]  (3.53)

where the coefficients of the pressure correction equation are the same as those of the pressure equation and the source term in the pressure correction equation \( b_{p'} \) is given by:

\[ b_{p'} = -u_e^* \Delta y\Delta z + u_w^* \Delta y\Delta z - v_n^* \Delta x\Delta z + v_s^* \Delta x\Delta z - w_t^* \Delta x\Delta y + w_b^* \Delta x\Delta y \]  (3.54)

### 3.3 Turbulence Modeling

The Reynolds-averaged Navier-Stokes equations are derived by averaging the viscous conservation laws over a time interval. Hence, all variables shown below are considered to be averaged quantities. The time averaged continuity equation is given as:

\[ \frac{\partial \rho \bar{v}}{\partial t} + \nabla \cdot (\rho \bar{v}) = 0 \]  (3.55)
In the absence of body forces, the turbulent mean momentum equation becomes:

\[
\frac{\partial \rho \bar{V}}{\partial t} + \nabla \cdot (\rho \bar{V} \otimes \bar{V} + p \bar{I} - \bar{\tau}^V - \bar{\tau}^R) = 0
\]  

(3.56)

where \( \otimes \) denotes the tensor product of the vectors \( \rho \bar{V} \) and \( \bar{V} \). The time averaged viscous shear stress is represented by \( \bar{\tau}^V \), and the Reynolds stress, \( \bar{\tau}^R \), in Cartesian coordinates based on Einstein's summation notation, is expressed as:

\[
\bar{\tau}^R_{ij} = -\rho \bar{v}'_i \bar{v}'_j
\]  

(3.57)

where \( \bar{v}'_i \) stands for the turbulent fluctuating velocity component. More detailed discussion of the turbulent averaging process can be found in Ref. [52]. Since the relations between the Reynolds stresses and the mean flow quantities are unknown, the application of the Reynolds-averaged equations to the computation of turbulent flows requires the introduction of some modeling of these unknown quantities to be added to the time averaged Navier-Stokes equations, based on theoretical and empirical considerations. This is a typical procedure which has been used for the last three decades to solve the so-called closure problems. Many different turbulence models, ranging from simple algebraic to multiple equation closure models, have been developed. The Cebeci-Smith model and the Baldwin-Lomax model are two famous zero-equation algebraic turbulence models which have been extensively used for thin, attached shear layers at moderate Mach numbers with very acceptable results [52]. However, as soon as the separation of the boundary layer is approached or shock wave appears, these algebraic models give rise to poor predictions. These deficiencies are related to theoretical limitations of the mixing length hypothesis, which implies that the eddy viscosity becomes zero if the mean velocity gradient vanishes (see Equation 3.58). This can lead to inconsistencies, in particular in the vicinity of separation or reattachment points. In addition, the transport and diffusion of turbulence cannot be taken into account by these algebraic models, and therefore the history effects is unable to be simulated.
Detailed discussion of the Cebeci-Smith model and the Baldwin-Lomax model are given in Appendix A.

### 3.3.1 TKE-$\epsilon$ Model

Because of the drawbacks of using the algebraic turbulence models mentioned previously, more sophisticated methods with higher-order accuracy have been developed based on transport equations for some basic turbulent properties such as the turbulent kinetic energy, \( TKE \), and the turbulent dissipation, \( \epsilon \). Although one-equation turbulence models have been developed for the TKE, the computational results were not considered sufficiently accurate. In the current research, we employ the well-known two-equation turbulence model, TKE-$\epsilon$ model, first to reflect the characteristics of turbulent transport and diffusion upon the numerical model, and then to generate the closed form solutions.

The Prandtl-Kolmogorov expression is given as:

\[
\nu_t = C_\nu (k^{1/2}) L \tag{3.59}
\]

where \( C_\nu \) is a constant, \( L \) and \( k^{1/2} \) act as the representative length scale and velocity of the turbulence, respectively (\( k \) is TKE). Also, from the dimensional arguments, the dissipation, \( \epsilon \), can be written as:

\[
\epsilon = C_\epsilon \left( \frac{k^{5/2}}{L} \right) \tag{3.60}
\]

and Equation 3.59 becomes:

\[
\nu_t = C_\mu \left( \frac{k^2}{\epsilon} \right) = \frac{\mu_t}{\rho} \tag{3.61}
\]

where \( C_\epsilon \) and \( C_\mu \) are constants.
The structures of these equations have the general form of a transport equation as shown in Equation 2.5. The TKE equation is written as:

$$\frac{\partial (pk)}{\partial t} + \frac{\partial (pk_u)}{\partial x} + \frac{\partial (pk_v)}{\partial y} + \frac{\partial (pk_w)}{\partial z} = \frac{\partial}{\partial x}\left(\frac{\mu_{eff} \partial k}{\sigma_k \partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu_{eff} \partial k}{\sigma_k \partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu_{eff} \partial k}{\sigma_k \partial z}\right)$$

$$+ (G - \rho \epsilon)$$  \hspace{1cm} (3.62)

and the dissipation equation is:

$$\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon u)}{\partial x} + \frac{\partial (\rho \epsilon v)}{\partial y} + \frac{\partial (\rho \epsilon w)}{\partial z} = \frac{\partial}{\partial x}\left(\frac{\mu_{eff} \partial \epsilon}{\sigma_\epsilon \partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu_{eff} \partial \epsilon}{\sigma_\epsilon \partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu_{eff} \partial \epsilon}{\sigma_\epsilon \partial z}\right)$$

$$+ \frac{\epsilon}{k} (C_1 G - C_2 \rho \epsilon)$$  \hspace{1cm} (3.63)

where

$$\mu_{eff} = \mu_0 + \mu_t$$

and

$$G = \nu_t \{2[(\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + (\frac{\partial w}{\partial z})^2] + [(\frac{\partial u}{\partial y}) + (\frac{\partial v}{\partial z})]^2 + [(\frac{\partial u}{\partial z}) + (\frac{\partial w}{\partial y})]^2 + [(\frac{\partial w}{\partial x}) + (\frac{\partial u}{\partial z})]^2\}$$

$$C_1 = 1.43; \ C_2 = 1.92; \ \sigma_\epsilon = 1.3; \ \sigma_k = 1.0$$  \hspace{1cm} (3.64)

### 3.3.2 Discretization of TKE and $\epsilon$ Equations

By regrouping the convective and the diffusive terms, the TKE equation can be written as:

$$\frac{\partial (pk)}{\partial t} + \frac{\partial}{\partial x}\left(\rho k u - \frac{\mu_{eff} \partial k}{\sigma_k \partial x}\right) + \frac{\partial}{\partial y}\left(\rho k v - \frac{\mu_{eff} \partial k}{\sigma_k \partial y}\right) + \frac{\partial}{\partial z}\left(\rho k w - \frac{\mu_{eff} \partial k}{\sigma_k \partial z}\right) = (G - \rho \epsilon)$$  \hspace{1cm} (3.66)

Integrated over the typical main control volume, the TKE equation becomes:

$$\int_t^{t+\Delta t} \int_0^L \int_0^W \int_0^H \left(\frac{\partial (pk)}{\partial t} + \frac{\partial}{\partial x}\left(\rho k u - \frac{\mu_{eff} \partial k}{\sigma_k \partial x}\right) + \frac{\partial}{\partial y}\left(\rho k v - \frac{\mu_{eff} \partial k}{\sigma_k \partial y}\right) + \frac{\partial}{\partial z}\left(\rho k w - \frac{\mu_{eff} \partial k}{\sigma_k \partial z}\right)\right) dx dy dz dt$$

$$= \int_t^{t+\Delta t} \int_0^L \int_0^W \int_0^H (G - \rho \epsilon) dx dy dz dt$$
which on rearrangement yields:

\[
\{(\rho k) - (\rho k)^0\}\left(\frac{\Delta x \Delta y \Delta z}{\Delta t}\right) + \alpha(J_{k_e} - J_{k_w} + J_{k_n} - J_{k_s} + J_{k_t} - J_{kb})
+ (1 - \alpha)(J_{e_0}^0 - J_{e_0}^o + J_{n_0}^0 - J_{s_0}^0 + J_{t_0}^0 - J_{b_0}^0)
= S_k(\Delta x \Delta y \Delta z)
\]

where \( J \) is the turbulent kinetic energy flux.

Similarly, the conservative form of the \( \epsilon \) equation can be rearranged by grouping the convective flux and the diffusive flux together as:

\[
\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial}{\partial x}(\rho \epsilon u - \frac{\mu_{eff}}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x}) + \frac{\partial}{\partial y}(\rho \epsilon v - \frac{\mu_{eff}}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial y}) + \frac{\partial}{\partial z}(\rho \epsilon w - \frac{\mu_{eff}}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial z}) = \frac{\epsilon}{k}(C_1 G - C_2 \rho \epsilon)
\]

(3.67)

Integrating the above equation over the main control volume, the dissipation equation becomes:

\[
\int_0^t \int_s^s \int_w^w \int_{b}^{e} \left[ \frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial}{\partial x}(\rho \epsilon u - \frac{\mu_{eff}}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x}) + \frac{\partial}{\partial y}(\rho \epsilon v - \frac{\mu_{eff}}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial y}) + \frac{\partial}{\partial z}(\rho \epsilon w - \frac{\mu_{eff}}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial z}) \right] dxdydzdt
= \int_t^0 \int_s^s \int_w^w \int_{b}^{e} \frac{\epsilon}{k}(C_1 G - C_2 \rho \epsilon) dxdydzdt
\]

The integral form of the \( \epsilon \) equation is represented as:

\[
\{(\rho \epsilon) - (\rho \epsilon)^0\}\left(\frac{\Delta x \Delta y \Delta z}{\Delta t}\right) + \alpha(J_{\epsilon_e} - J_{\epsilon_w} + J_{\epsilon_n} - J_{\epsilon_s} + J_{\epsilon_t} - J_{\epsilon_b})
+ (1 - \alpha)(J_{\epsilon_0}^0 - J_{\epsilon_0}^o + J_{\epsilon_0}^0 - J_{\epsilon_s}^0 + J_{\epsilon_t}^0 - J_{\epsilon_b}^0)
= S_\epsilon(\Delta x \Delta y \Delta z)
\]

where \( J \) represents the turbulent dissipation flux.

Using the same methodology illustrated in discretizing the momentum equations, the final discretized algebraic equations for the turbulent kinetic energy and turbulent dissipation are obtained as:
For the TKE equation:

\[
\tilde{a}_{kP} k_P = \sum (\tilde{a}_{knb} k_{nb}) + b_k + S_k (\Delta x \Delta y \Delta z) \quad (3.68)
\]

where

\[
\tilde{a}_{kE} = \alpha \cdot a_{kE} : \quad \tilde{a}_{kW} = \alpha \cdot a_{kW} \\
\tilde{a}_{kN} = \alpha \cdot a_{kN} : \quad \tilde{a}_{kS} = \alpha \cdot a_{kS} \\
\tilde{a}_{kT} = \alpha \cdot a_{kT} : \quad \tilde{a}_{kB} = \alpha \cdot a_{kB}
\]

\[
\tilde{a}_{kP} = \frac{\rho \Delta x \Delta y \Delta z}{\Delta t} + \alpha (a_{kE} + a_{kW} + a_{kN} + a_{kS} + a_{kB} + a_{kT})
\]

\[
b_k = (1 - \alpha) [a_{kE}^0 k_E^0 + a_{kW}^0 k_W^0 + a_{kN}^0 k_N^0 + a_{kS}^0 k_S^0 + a_{kT}^0 k_T^0 + a_{kB}^0 k_B^0]
\]

\[
- (a_{kE}^0 + a_{kW}^0 + a_{kN}^0 + a_{kS}^0 + a_{kT}^0 + a_{kB}^0) k_P^0
\]

\[
+ \left( \frac{\rho \Delta x \Delta y \Delta z}{\Delta t} \right) k_P^0 + S_k \Delta x \Delta y \Delta z
\]

For the \( \epsilon \) equation:

\[
\tilde{a}_{\epsilon P} \epsilon_P = \sum (\tilde{a}_{\epsilon nb} \epsilon_{nb}) + b_\epsilon + S_\epsilon (\Delta x \Delta y \Delta z) \quad (3.69)
\]

where

\[
\tilde{a}_{\epsilon E} = \alpha \cdot a_{\epsilon E} : \quad \tilde{a}_{\epsilon W} = \alpha \cdot a_{\epsilon W} \\
\tilde{a}_{\epsilon N} = \alpha \cdot a_{\epsilon N} : \quad \tilde{a}_{\epsilon S} = \alpha \cdot a_{\epsilon S} \\
\tilde{a}_{\epsilon T} = \alpha \cdot a_{\epsilon T} : \quad \tilde{a}_{\epsilon B} = \alpha \cdot a_{\epsilon B}
\]

\[
\tilde{a}_{\epsilon P} = \frac{\rho \Delta x \Delta y \Delta z}{\Delta t} + \alpha (a_{\epsilon E} + a_{\epsilon W} + a_{\epsilon N} + a_{\epsilon S} + a_{\epsilon T} + a_{\epsilon B})
\]

\[
b_\epsilon = (1 - \alpha) [a_{\epsilon E}^0 \epsilon_E^0 + a_{\epsilon W}^0 \epsilon_W^0 + a_{\epsilon N}^0 \epsilon_N^0 + a_{\epsilon S}^0 \epsilon_S^0 + a_{\epsilon T}^0 \epsilon_T^0 + a_{\epsilon B}^0 \epsilon_B^0]
\]

\[
- (a_{\epsilon E}^0 + a_{\epsilon W}^0 + a_{\epsilon N}^0 + a_{\epsilon S}^0 + a_{\epsilon T}^0 + a_{\epsilon B}^0) \epsilon_P^0
\]

\[
+ \left( \frac{\rho \Delta x \Delta y \Delta z}{\Delta t} \right) \epsilon_P^0 + S_\epsilon \Delta x \Delta y \Delta z
\]

The \( a_P, a_{nb} \) and \( b \) are respectively the coefficient of the primary grid, the coefficients of the neighboring auxiliary grids and the source term with respect to the main control.
volume, which is the same as the control volume used for the pressure or pressure correction equation. All the coefficient expressions are similar to those of the momentum equations. The only difference is that the value of diffusivity, \( \Gamma \), is \( \mu_{eff} \) in the momentum equations and it changes to \( \mu_{eff}/\sigma_k \) for the TKE equation or \( \mu_{eff}/\sigma_\epsilon \) for the dissipation equation.

### 3.4 Wall Function

The TKE-\( \epsilon \) model presented above is often referred to as a high Reynolds number model which is valid only for a fully turbulent flow. The local Reynolds number of turbulence is given as:

\[
Re_t = (k^{1/2}/l)/\nu
\]

(3.70)

where

\[
l = (k^{3/2})/\epsilon
\]

(3.71)

or

\[
Re_t = \frac{k^2}{\nu\epsilon}
\]

(3.72)

It is clear that \( Re_t \) is proportional to the ratio of the eddy viscosity to the molecular viscosity. However, close to solid walls and some other interfaces, there are inevitably regions where the local Reynolds number of turbulence is so small that viscous effects dominate over the turbulent ones, and the molecular viscosity may be of the same order or larger than the turbulent viscosity, rendering the TKE-\( \epsilon \) model invalid. There are two methods of accounting for these regions in numerical processes for computing the turbulent flow: the low-Reynolds-number model and the wall-function method. Basically, the methodology of the low-Reynolds-number method is to devise the turbulence-model equations to be applicable throughout the laminar, semi-laminar, and fully turbulent regions. The details regarding implementation of the low-Reynolds-number model can
be found in Ref. [53]. In this study, the wall-function method was employed for the following two reasons: it economizes computer time and storage, and it allows the introduction of additional empirical information in special cases, such as when the wall is rough. Consider the adjacent grid points $W$ and $P$ of a finite-difference grid on which the flow field is to be analyzed (see Figure 3.3). It is important to ensure that when using the wall-function method, point $P$ is sufficiently remote from point $W$, which lies on the wall, for $Re_t$ to be much greater than unity; so much greater in fact, that the viscous effects are entirely overwhelmed there by the turbulent ones. The definitions of shear stress on the wall and of the friction velocity are quoted here for convenience:

$$\tau_w = \mu_t \left( \frac{\partial u}{\partial y} \right)$$

$$u_r = \sqrt{\frac{\tau_w}{\rho}}$$

The following two algebraic equations are used to calculate the turbulent kinetic energy and the turbulent dissipation in the applicable flow region:

$$k = \frac{u_r^2}{\sqrt{C_\mu}} \quad (3.73)$$

$$\epsilon = \frac{u_r^3}{(\kappa y)} \quad (3.74)$$

Therefore, the turbulent viscosity can be found by using the equation shown below:

$$\mu_t = \rho \cdot C_\mu \cdot \left( \frac{\kappa^2}{\epsilon} \right) \quad (3.75)$$

where $C_\mu$ is 0.09 and $\kappa$ is 0.42. The flow region is applicable where the wall-function method is chosen by iteratively estimating the physical length $y$ and then calculating the local Reynolds number there until $Re_t$ is much greater than unity. Therefore, depending on the detail necessary, a coarser grid can be used in this near-wall flow field for economizing computational time and storage. In Chapter 5, a two-dimensional channel flow problem will be used to illustrate the guidelines for using the wall-function method.
3.5 Solution Procedure-SIMPLER Algorithm

The solution procedure for solving the discretized governing equations based on the SIMPLER algorithm (Semi-Implicit Method for Pressure-Linked Equations) [51] is given below:

STEP 1: Guess the velocity fields \( u, v \) and \( w \).

STEP 2: Calculate \( \hat{u}, \hat{v} \) and \( \hat{w} \) pseudo-velocity fields using Equation 3.24 through Equation 3.29.

STEP 3: Calculate the source term of the pressure equation, \( b_p \), using Equation 3.31 and then solve Equation 3.30, the pressure equation, to obtain the pressure field, \( p \).
STEP 4: Using \( p \), calculate the source terms of the momentum equations, and then solve Equations 3.8, 3.9 and 3.10 to obtain the uncorrected velocity fields \( u' \), \( v' \) and \( w' \).

STEP 5: Calculate the source term of the pressure correction equation, \( b_{p'} \), using Equation 3.54, and then solve Equation 3.53, the pressure correction equation, to obtain the pressure correction field, \( p' \).

STEP 6: Using \( p' \), calculate the velocity correction fields \( u' \), \( v' \) and \( w' \) from Equations 3.41, 3.42 and 3.43.

STEP 7: Update the velocity fields using Equations 3.35, 3.36 and 3.37.

STEP 8: Calculate the source terms of the TKE equation and \( \epsilon \) equation using Equation 3.64 and Equation 3.65.

STEP 9: Solve Equations 3.68 and 3.69 to obtain the TKE and \( \epsilon \) distributions in the internal numerical domain, and then combine the information of TKE and \( \epsilon \) in the near-wall region obtained from Equations 3.73, 3.74 and 3.75 using the wall-function method.

STEP 10: Return to STEP 2 and repeat until convergence is obtained.

STEP 11: Calculate the other flow quantities, such as pressure coefficients on the body.

3.6 Higher-Order Accuracy Schemes

3.6.1 Motivation

The SIMPLE [54] family of algorithms (SIMPLER [51], SIMPLC [55]) has been very popular with the introduction of the Power-Law approximation to the convection-diffusion equation. The strength of the Power-Law scheme, based on first-order upwind, is its diagonally dominant discretized equations with excellent convergence properties.
and predictable, physically smooth solutions. However, for high Reynolds numbers flows with turbulent characteristics, the robustness of the first-order upwind approximation is severely handicapped by artificial diffusion obscuring the natural viscous processes. With a desire to reduce the false diffusion characteristic of the first-order upwind scheme and to maximize accuracy, several higher-order upwind schemes have been introduced [56,57]. Routine industry use of the higher-order schemes is not the order as yet attributable in part to the numerical instability traceable to the nonphysical oscillations in the regions of steep gradients.

A popular approach to remedy some of the stability problems of the higher-order upwind schemes is to use the first-order upwind scheme as the basis, and assign the additional terms from the higher-order treatment to the source terms of the discretized equations to be treated explicitly. This simple and yet powerful technique retains the diagonal dominance of the discretized algebraic equations during the iterative updating without compromising the higher-order accuracy. Most of the procedures outlined in the literature [58-63] for casting the higher-order schemes into special forms discuss the details only from the framework of uniform grid.

By using Lagrange interpolation, which is suitable for the construction of higher-order schemes of any order, to represent the interface convection terms, this study introduces a technique for avoiding the misinterpretation of turbulent flow under high Reynolds number conditions that occur when using the Power-Law scheme. This is the direct result of Power-Law scheme neglecting the viscous terms when the absolute value of the Peclet number is greater than ten. Two higher-order schemes, namely, second-order upwind (SOU) [64,65] and QUICK [66], are chosen to demonstrate the formulation for uniform and non-uniform grid spacings without loss of generality. Furthermore, the properties of Lagrange interpolation are exploited to cast the higher-order discretized equations as a combination of first-order terms and source terms in an efficient and natural manner. Use of Lagrange interpolation thus paves the way for algebraic manipulation of the higher-
order schemes of any order, and for strict compliance with the Scarborough criterion as laid out by Patankar [51]. Detailed derivations of SOU and QUICK schemes based on Lagrange interpolation are given in Appendix B.

Finally, there has been an incessant interest in the research community in finding the numerical simulation procedures that intelligently switch between schemes, depending on the local flow conditions, in order to retain the best properties of the various schemes. Despite improved results, widespread use of these techniques is lacking due to their inherent complexity. In this study, implementing the Power-Law scheme and higher-order schemes such as SOU and QUICK together by switching from one to another based on the cell Reynolds number (Peclet number), is a newly developed approach to improve the poor stability characteristics which, in general, higher-order schemes inherit, without loss of computational accuracy.

3.6.2 QUICK Scheme

This scheme estimates the interface value by interpolating two upwind biased neighbors of the interface and one downstream of the interface. Using the Lagrange interpolation, the final formulation of the three-dimensional discretized convection-diffusion equation in the QUICK scheme can be written as:

\[ ap\partial p = a_{E}\partial_{E} + a_{W}\partial_{W} + a_{N}\partial_{N} + a_{S}\partial_{S} + a_{T}\partial_{T} + a_{B}\partial_{B} + b \]  

(3.76)

where

\[ a_{E} = D_{e} + [-F_{e}, 0] \]  

(3.77)

\[ a_{W} = D_{w} + [F_{w}, 0] \]  

(3.78)

\[ a_{N} = D_{n} + [-F_{n}, 0] \]  

(3.79)

\[ a_{S} = D_{s} + [F_{s}, 0] \]  

(3.80)

\[ a_{T} = D_{t} + [-F_{t}, 0] \]  

(3.81)
\[ a_B = D_b \pm \{ F_b \pm 0 \} \]  
(3.82)

\[ a_P = a_E + a_W + a_N + a_S + a_T + a_B \]

\[-(\gamma_{1e}\{ F_e \pm 0 \} + \delta_{2e}\{ -F_e \pm 0 \}) \]

\[-(\gamma_{1n}\{ F_n \pm 0 \} + \delta_{2n}\{ -F_n \pm 0 \}) \]

\[-(\gamma_{1f}\{ F_f \pm 0 \} + \delta_{2f}\{ -F_f \pm 0 \}) \]  
(3.83)

and

\[ b = \{ \delta_{1e}O_E + \delta_{2e}O_{EE} - (\delta_{1e} + \delta_{2e})O_E \} \{ -F_e \pm 0 \} \]

\[-\{ \gamma_{1e}O_W + \gamma_{2e}(O_E - O_P) \} \{ F_e \pm 0 \} \]

\[ + \{ \gamma_{1w}O_W + \gamma_{2w}O_P - (\gamma_{1w} + \gamma_{2w})O_W \} \{ F_w \pm 0 \} \]

\[-\{ \delta_{1w}(O_W - O_P) + \delta_{2w}O_E \} \{ -F_w \pm 0 \} \]

\[ + \{ \delta_{1n}O_P + \delta_{2n}O_{NN} - (\delta_{1n} + \delta_{2n})O_N \} \{ -F_n \pm 0 \} \]

\[-\{ \gamma_{1n}O_S + \gamma_{2n}(O_N - O_P) \} \{ F_n \pm 0 \} \]

\[ + \{ \gamma_{1s}O_S + \gamma_{2s}O_P - (\gamma_{1s} + \gamma_{2s})O_S \} \{ F_s \pm 0 \} \]

\[-\{ \delta_{1s}(O_S - O_P) + \delta_{2s}O_N \} \{ -F_s \pm 0 \} \]

\[ + \{ \delta_{1t}O_P + \delta_{2t}O_{TT} - (\delta_{1t} + \delta_{2t})O_T \} \{ -F_t \pm 0 \} \]

\[-\{ \gamma_{1t}O_B + \gamma_{2t}(O_T - O_P) \} \{ F_t \pm 0 \} \]

\[ + \{ \gamma_{1b}O_B + \gamma_{2b}O_P - (\gamma_{1b} + \gamma_{2b})O_B \} \{ F_b \pm 0 \} \]

\[-\{ \delta_{1b}(O_B - O_P) + \delta_{2b}O_T \} \{ -F_b \pm 0 \} \]  
(3.84)

where \( \gamma \)'s and \( \delta \)'s are the geometric coefficients which are expressed by the following relations based on the index system shown in Figure 3.4:

\[ \gamma_1 = \frac{x_{i-\frac{1}{2}} - x_{i-1}}{x_i - x_{i-1}} \left( \frac{x_{i-\frac{1}{2}} - x_{i-2}}{x_i - x_{i-2}} \right) \quad \gamma_2 = \frac{x_{i-\frac{1}{2}} - x_{i-1}}{x_{i-2} - x_{i-1}} \left( \frac{x_{i-\frac{1}{2}} - x_{i}}{x_{i-2} - x_i} \right) \]
The coefficients along the $y$ or $z$ direction can be obtained by changing $x$ to $y$ or $z$ and $i$ to $j$ or $k$, respectively.

### 3.6.3 SOUM and QUICKM Schemes

In Figure 3.5, the Power-Law scheme shows excellent agreement with the exact solution if the absolute value of the grid Peclet number is less than 6 [51]. This implies that the Power-Law scheme is capable of providing more accurate information when the grid Peclet number is located within this specific range. Therefore, a judicious combination of the SOU or QUICK schemes with the Power-Law scheme will be the optimal choice, not only to obtain a correct prediction, but also to preserve the stability and convergence characteristics required for all numerical simulations. The flow chart shown in Figure 3.6 highlights the concept for implementing the SOUM or QUICKM schemes. Hereafter QUICK and QUICKM schemes with higher order accuracy are used for all numerical simulations.
Figure 3.5  Relationship of Peclet number and $A(|P|)$ function [51]
Figure 3.6  Flow chart for using SOUM scheme or QUICKM scheme
Qualitative results from field observations of a living-tree shelterbelt under real atmospheric flow conditions, as well as a wind-tunnel flow visualization of scale-model fences were used to explore the fundamental phenomena of shelterbelt flow to help in the numerical modeling. Detailed procedures for these two experimental approaches are presented in the following sections.

4.1 Field Observation

This series of the field experiments was the result of a cooperative effort among the University of Nebraska, the Rocky Mountain Forest and Range Experiment Station, and Iowa State University. The data shown here, including the vertical windspeed profiles, temperature distribution, wind direction at one level, relative humidity, and the static pressure differential across the living-tree shelterbelt, are obtained from two field observations that took place from Sep. 21-28, 1993 and May 8-15, 1994. The field site is located approximately 50 km north of Lincoln, NE near Mead at 41°10' North Latitude and 96°40' West Longitude.

The broad objectives of this series of field experiments are to improve understanding of the aerodynamics of shelterbelts, and to apply that information in predicting the economic impact of climate change on croplands protected by shelterbelts. The immediate goal of the first experiment is to explore the capabilities of the data acquisition system at this experimental site. Figure 4.1 shows the first measurement transect within the
experimental farm. Mast No. 1 was centered in a vegetable plot and remained at the same location throughout the experiment for recording the freestream airflow conditions. Masts No. 2 and No. 3 were moved across the alfalfa plot between the two east-west legs of the shelterbelt array for catching the aerodynamic data in the vicinity of the shelterbelt. The belts consist of two rows, 3 meters apart, of alternating pairs of green ash and white pine. During the measurement period the average width and height of the shelterbelt are 8 meters and 12 meters, respectively. Detailed information is shown in Figure 4.2. Each instrument mast has arms holding cup anemometers for measuring the windspeed and thermistors for measuring the relative humidity at ten levels up to a 10-meter height. The thermistor attaches midway between the anemometer and the mast at each level. There is one propeller-vane near the 4-meter level and a probe in a radiation shield mounted on the back brace of each mast for measuring the wind direction, temperature and relative humidity at a height of 1.5 meters. The static pressure differential across the shelterbelt is measured by a pressure transducer. Figure 4.3 is a sketch of the observation mast.

The objectives of the second experiment are to record vertical profiles of the windspeed and temperature near a two-row eastern red cedar windbreak, to measure the gradients of static base pressure near the shelterbelt, and to begin measurements of wind and temperature profiles near the riparian sites. Figures 4.4 and 4.5 show the measurement transects and the dimension of the second experiment farm, respectively. The shelterbelt is formed by two rows of red cedar with an average height of 4.5 meters. The same equipment and similar procedures as the first experiment are utilized to obtain the aerodynamic information near this shelterbelt.

In the interest of brevity, only some of the observed and analyzed results are illustrated. The details of the experimental site, apparatus, procedures, observation data and the data analyses are given in the reports [67-69]. In Figures 4.6 and 4.7, the positions of three observation masts and the vertical windspeed profile of each mast from set
No. 6 to set No. 10 of the first experiment are chosen to demonstrate the observation sequence and selected results of the field investigations. The windspeed distribution and the wind reduction percentage of set No. 21 from the second experiment are shown in Figure 4.8. It is clearly seen that the approximate logarithmic velocity profile in the upstream undisturbed flow region is captured, and a large amount of windspeed reduction is observed close to either the windside or lee side of the shelterbelt.

Figure 4.9 shows the static pressure distribution across the shelterbelt of the second experiment conducted in 1994. Three different pressure gradient regions were found: a positive pressure gradient in the windward (from -1H to -1H), a deep pressure drop across the shelterbelt (-1H to +2H), and a gradual recovery region far downstream (+2H to +11H), where H is the height of the shelterbelt. It is this pressure distribution which causes the approaching oblique wind to orient parallel to the shelterbelt in the near windside, and then, perpendicularly to the belt as soon as the airflow contacts the leading edge of the shelterbelt. Finally, the wind direction in the far leeside gradually changes back to the upstream wind direction. The schematic representation of wind direction across the shelterbelt is shown in Figure 4.10.

Two observed data sets, No. 1 and No. 53 from the second field experiment, illustrating the wind direction distributions in the shelterbelt flowfield, are selected to describe the phenomena mentioned above (see Figures 4.11). According to the data regression analysis accomplished by Alkhalil [4], the following conclusions can be drawn: a significant linear relationship with a positive slope is found at the lower level (below 0.5H) between the angle of incidence, which is defined as the angle between the approaching wind and the direction normal to the shelter, and the windspeed reduction in the lee. This is because the increase in the distance travelled by the airflow across the shelterbelt results in an increase in frictional forces between the shelter elements and the air particles.

The data also reveal a positive linear relationship between the windspeed in the open
and the reduced windspeed in the leeward of the shelterbelt. A higher windspeed of the undisturbed flow creates a larger pressure gradient between the windside and leeside of the shelterbelt. This significant amount of pressure drop imposes more deflection on the flow, which in turn increases the frictional forces as the airflow moves through the shelterbelt, so as to increase the sheltering effect behind the belt.

Figure 4.1 Transect of the first field observation [67] (courtesy of Dr. Schmidt)
Figure 4.2  First field experiment setup

September 21-28, 1993
Figure 4.3 Sketch of the observation mast
Figure 4.4 Transect of the second field observation [68] (courtesy of Dr. Schmidt)
Figure 4.5  Second field experiment setup
Figure 4.6  Positions of three observation masts from set No. 6 to set No. 10 of the first experiment [67] (courtesy of Dr. Schmidt)
Figure 4.7 Vertical windspeed profile of each mast from set No. 6 to set No. 10 of the first experiment [67] (courtesy of Dr. Schmidt)
Figure 4.8  Windspeed distribution and wind reduction percentage of set No. 21 of the second experiment
Pressure Coefficient

<table>
<thead>
<tr>
<th>Location w/r wind travel (H)</th>
<th>-0.8</th>
<th>-0.6</th>
<th>-0.4</th>
<th>-0.2</th>
<th>0</th>
<th>0.2</th>
<th>0.4</th>
</tr>
</thead>
<tbody>
<tr>
<td>10-11 May South wind</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13 May Southwest wind</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 1.9 Static pressure distribution across the shell depth of the second experiment [68] (courtesy of Dr. Schmidt)
Figure 4.10  Schematic diagram of wind direction across the shelterbelt
Figure 4.11 Wind direction distributions in the shelterbelt flowfield, sets No. 1 and No. 55 from the second field experiment.
4.2 Flow Visualization

A series of experiments for studying the airflow passing through two-dimensional and three-dimensional scaled porous fences has been conducted by using the Rockwell Collins 960-1 smoke tunnel, shown in Figure 4.12, in the Department of Aerospace Engineering and Engineering Mechanics at Iowa State University in May, 1994. Videotaping and photography provide static and dynamic techniques for visualizing the flow patterns surrounding the scaled models. The test section of the smoke tunnel is 37 inches by 24 inches with a 2.5-inch depth. Paraffinic oil is used to generate the smoke. Two sets of scaled models have been constructed: two-dimensional fences 12 inches by 2.5-inches (aspect ration = 4.8) and three-dimensional fences 7.25 inches by 1.5 inches (aspect ration =4.83). Each model set, which is made of cardboard for easy cutting, has different porosities ranging from 0 % to 60% in 10 % increments. The reason the two or three dimensional models are defined is that the depth of the test section is the same as the depth of the two-dimensional model so that the airflow can either move across the tip of the model or through the model if the model is porous. In contrast, in the three-dimensional model, the air not only can move across or through, but it can also go over the top of the fences since the height of the three-dimensional model is smaller than that of the test section.

The focus of this series of tests is to measure the size of the wake behind the fence at different orientations of the model, but with fixed running conditions. Figure 4.13 displays a series of pictures showing the streamline contours surrounding the 20% porosity two-dimensional model from 0 degrees to 70 degrees with respect to the direction normal to the freestream. The wake region behind the model is clearly seen. The approaching flow tends to orient parallel to the fence and to penetrate the model perpendicularly. This phenomenon is consistent with both observations from the field experiment [4], and the findings from wind tunnel tests [70]. Quantitative comparisons of the wake width
between the two-dimensional and three-dimensional models are shown from Figures 4.14 to 4.25. Results of this series of experiments can be summarized as follows: (1) The wake width decreases, which implies that the pressure drag decreases, when the porosity of the model increases. This situation prevails over all the running cases, both for two-dimensional and three-dimensional models. (2) For each model with a certain porosity, the wake width decreases when the model becomes more parallel to the approaching flow. Since the thickness of the model is relatively small (0.125 inches), the friction effect induced by the air particles penetrating the fence is less significant compared to the case of living-tree shelterbelts. Therefore, the wind reduction in the near leeside is not strongly correlated with the orientation of these scaled fences. (3) If two comparative models have the same porosity and the same orientation, due to three-dimensional relieving effect, the wake width in the lee of the three-dimensional model is always less than the wake width behind the two-dimensional one.

Figure 4.12 ISU smoke tunnel (courtesy of Rockwell Collins)
Figure 4.13 Streamline contours surrounding the 20% porosity two-dimensional model with 0 to 70 degrees approach angle.
Figure 4.14  Comparisons of the wake width at 0 degrees approach angle

Figure 4.15  Comparisons of the wake width at 10 degrees approach angle
Figure 4.16  Comparisons of the wake width at 30 degrees approach angle

Figure 4.17  Comparisons of the wake width at 50 degrees approach angle
Figure 4.18  Comparisons of the wake width at 70 degrees approach angle

Figure 4.19  Comparisons of the wake width at 0% porosity
Figure 4.20  Comparisons of the wake width at 10% porosity

Figure 4.21  Comparisons of the wake width at 20% porosity
Figure 4.22  Comparisons of the wake width at 30% porosity

Figure 4.23  Comparisons of the wake width at 40% porosity
Figure 4.24  Comparisons of the wake width at 50% porosity

Figure 4.25  Comparisons of the wake width at 60% porosity
5 TWO-DIMENSIONAL AND THREE-DIMENSIONAL TEST CASES

Before results of shelterbelt simulations are presented, a comparative analysis of the higher order schemes evaluated are presented.

5.1 Two-Dimensional Test Cases

5.1.1 2-D Driven-Cavity Flow

A two-dimensional circulating flow of a fluid in a square cavity with three bounded sides and the top wall moving with constant speed is the first test problem chosen to assess the performance of the SOU, QUICK, SOUM and QUICKM schemes for modeling the convective kinematics of the convection-diffusion equation. Three cases with Reynolds numbers ranging from 400 to 3200 and three levels of grids, namely, 22 by 22, 42 by 42 and 82 by 82, are studied. The numerical results with the 22 by 22 non-uniform grids at Reynolds numbers 400 and 1000 are presented in this section. The results from the other running conditions are shown in Appendix B.

The following simple algebraic equation is used to generate the smoothly varying grid:

\[ X_i = X_0 + \left\{ \left( \frac{i - 1}{i_{end} - 2} \right) \right\} \cdot L \]  

(5.1)

where \( i_{end} \) is the total number of grid points and \( i \) is the index varying from 1 to \( i_{end} \). \( X_i \) is the location on station number \( i \). \( X_0 \) is the origin and \( L \) is the total length of the
numerical domain. Stretching of the grid format is controlled by a parameter, \( a \). For a uniform grid, \( a \) is unity, and it is greater than one for a non-uniform grid. As \( a \) increases, the grid spacing becomes finer near the origin and becomes more uniform away from the origin. For the 22 by 22 non-uniform grid spacing, \( L \) is 0.5. \( i_{end} \) is 12. \( i \) runs from 1 to 12. \( x_0 \) is zero and \( a \) is 2. We then use symmetry to generate the other half of the numerical domain. Figure 5.1 shows the physical domain with boundary conditions for the two-dimensional driven cavity problem, and the computational domain with the 22 by 22 grid is shown in Figure 5.2.

For all cases, fine grid solutions obtained using central difference, vorticity-stream function approach with the multigrid, by Ghia [71] are used as the benchmark. The results from the Power-Law scheme are also presented for illustrating the differences between the higher-order and lower-order schemes. Figures 5.3 and 5.4 show the horizontal velocity distributions along the vertical centerlines for the 22 by 22 non-uniform grid for the Reynolds numbers 400 and 1000. It clearly shows that there are three categories of solution: SOU and QUICK schemes form one category, SOUM and QUICKM schemes belong to a second, and the third one is the Power-Law scheme. The results from the Power-Law scheme always show underestimation with respect to the reference solutions, and the deviation becomes larger as the Reynolds number increases. The predictions using methods from the second category are always between those from the first and the third categories. For low Reynolds number cases, SOU and QUICK schemes can accurately model the centerline velocity distribution; however, for higher Reynolds number cases, the results obtained by using these higher-order schemes display significant overshoots compared to the benchmarks. In contrast, SOUM and QUICKM schemes show better agreement with the reference solution for higher Reynolds number cases. It is observed that SOUM and QUICKM schemes start deviating from the Power-Law scheme at \( Re=400 \) and begin approaching the SOU and QUICK schemes at higher Reynolds number cases. This is because the range of Peclet numbers for the
low Reynolds number case is much smaller than that of the high Reynolds number case: the region eligible for using the Power-Law scheme in the numerical domain becomes smaller as the Reynolds number increases.

The plots of convergent history (overall residual versus iteration number) for the cases of Reynolds number 400 and 1000 are given in Figures 5.5 and 5.6. The definition of the overall residual is given as follows:

\[
\sqrt{\left\{ \sum (a_P^u u_P - (\sum a_{n_b}^u u_{n_b} + b_u))^2 + \sum (a_P^v v_P - (\sum a_{n_b}^v v_{n_b} + b_v))^2 \right\} / N}
\]  (5.2)

where superscription \( u \) and \( v \) denote the coefficients belonging to the \( u \) and \( v \) momentum equations, respectively. \( P \) is the main grid, and subscript \( n_b \) stands for the neighboring grids with respect to \( P \). \( N \) is the total number of grid cells in the computational domain.

It is observed that the Power-Law scheme preserves better computational stability and displays excellent convergence characteristics for all studied cases. As with all higher-order schemes, SOU and QUICK schemes do not perform as well as the Power-Law scheme in terms of convergence characteristics. In most cases, QUICK shows better convergence characteristics than the SOU scheme. For lower Reynolds numbers and coarse grid cases, the SOUM and QUICKM schemes exhibit the best convergent history compared to the SOU and QUICK schemes, especially, for a Reynolds number of 400 for the 22 by 22 grid spacing case. This benefit diminishes as the Reynolds number becomes higher. Again, this is due to the reduced involvement of the Power-Law scheme when implementing these combination schemes as the cell Reynolds number increases. It is concluded that for the lower Reynolds number and coarse grid case, which is the most stable numerical testing condition and the one found to be most economical in terms of CPU time, the SOUM and QUICKM schemes are capable of providing accurate predictions and displaying better stability characteristics. In this study, the Power-Law scheme is used to obtain the preliminary information from the flow field, and then the QUICKM scheme is employed to analyze the shelterbelt aerodynamics.
Figure 5.1  Physical domain of a two-dimensional driven-cavity flow problem

Figure 5.2  Computational domain of a two-dimensional driven-cavity flow with a 22 by 22 non-uniform grid
Figure 5.3  U-velocity profile on the vertical centerline of the driven-cavity at Re=400 with a 22 by 22 non-uniform grid

Figure 5.4  U-velocity profile on the vertical centerline of the driven-cavity at Re=1000 with a 22 by 22 non-uniform grid
Figure 5.5 Convergence history at Re=400 with a 22 by 22 non-uniform grid for the driven-cavity flow.

Figure 5.6 Convergence history at Re=1000 with a 22 by 22 non-uniform grid for the driven-cavity flow.
5.1.2 2-D Backward-Facing Step Flow

A two-dimensional backward facing step flow with a parabolic inlet velocity profile is used as the second test problem. The physical domain is shown in Figure 5.7. In order to capture the velocity profile in the boundary layer, a finer grid is used in the middle region of the channel (see Figure 5.8). The computational domain is discretized by 102 uniform grid horizontally and 102 stretched grid vertically. Test conditions change from Reynolds number 200 to 800, increasing by increments of 200. Streamline contours using the QUICKM scheme for four cases are shown in Figure 5.9. A first-separation bubble behind the step is successfully captured. A second-separation region on the top wall is observed as the Reynolds reaches 600, which is consistent with the results obtained by Kim and Moin [72]. Figure 5.10 shows the comparison of the extension of the first-separation region among the QUICK, QUICKM and Power-Law schemes. The numerical prediction by Kim and Moin found by using the fractional-step method, and both experimental results and computational solution by using the TEACH scheme reported by Armaly et al. [73], are also shown in order to provide a reference solution. Again, it is evident that the Power-Law scheme is only sufficient to provide correct information in lower Reynolds number cases. On the other hand, the QUICK and QUICKM schemes display good agreement with the reference numerical results at higher Reynolds numbers as well.

As the Reynolds number approaches 600, due to the three-dimensional nature of the experimental flow under such flow conditions, numerical predictions start to deviate from the experimental values. [73]. Comparison of the size of the secondary separation bubble is displayed in Figure 5.11. Underestimation of the numerical results using the Power-Law scheme is expected, and good agreement between the high-order schemes and the reference solution are reaffirmed. The plots of convergent history for Reynolds numbers ranging from 400 to 800 are given in Figures 5.12 to 5.15. The Power-Law
scheme shows outstanding convergent characteristics for all cases, and the QUICKM scheme displays better convergent history than does QUICK, especially at Re=800. Figures 5.16 and 5.17 compare the horizontal velocity distributions among the Power-Law, QUICK and QUICKM schemes. The results obtained from Gartling [74] using the finite element method are also shown for comparison. At seven channel heights downstream, the prediction from the Power-Law scheme is significantly off from the reference data, and QUICKM performs much better than the QUICK scheme. There is no distinction between the QUICK and QUICKM schemes for the u-velocity profile at the x=15 channel heights location. In contrast, the Power-Law scheme underestimates the maximum velocity at the center and overpredicts the velocity closer to the walls.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{figure5.7}
\caption{Physical domain of a two-dimensional backward-facing step}
\end{figure}
Figure 5.8  Computational domain of a two-dimensional backward-facing step
Figure 5.9  Streamline patterns of a backward-facing step for $Re=200$, 400, 600 and 800
Figure 5.10 Dimension of the first separation region for the backward-facing step flow

Figure 5.11 Dimension of the secondary separation region for the backward-facing step flow
Figure 5.12 Convergence history at Re=200 for the backward-facing step flow

Figure 5.13 Convergence history at Re=400 for the backward-facing step flow
Figure 5.14 Convergence history at Re=600 for the backward-facing step flow

Figure 5.15 Convergence history at Re=800 with a 102 by 102 non-uniform grid spacing for a backward-facing step flow
Figure 5.16  U-velocity profile at x=7 channel heights for a backward-facing step flow at Re=800

Figure 5.17  U-velocity profile at x=15 channel heights for a backward-facing step flow at Re=800
5.1.3 2-D Flat-Plate Flow

The third two-dimensional test problem used is the flow moving along a flat-plate. The Power-Law, QUICK and QUICKM schemes are used to simulate the flow field. Both laminar and turbulent cases are tested. Figure 5.18 shows the physical domain and the boundary conditions. The computational domain with a 102 by 46 non-uniform grid is shown in Figure 5.19. The wall function is implemented within the flow region close to the wall, and the TKE-ε model is used to simulate the turbulent eddy diffusivity in the flow region far from the solid boundary. A no-slip condition is applied to the flat-plate, with a prescribed inlet condition. The top boundary of the numerical domain is a free boundary. At the outlet boundary, the flow velocity is determined from that at the next inner parallel plane by scaling the latter such that the overall mass flow is conserved.

Figure 5.20 depicts the laminar velocity profiles obtained from the Power-Law, QUICK and QUICKM schemes and compares them to the theoretical Blasius solution at Re=10⁴. In the same figure, the velocity profiles of the turbulent boundary layer obtained by using the TKE-ε model in the QUICK and QUICKM schemes are also presented. The turbulent velocity profile has a much steeper slope at the wall. This implies that shearing stress on the wall for the turbulent boundary layer is much larger than that for the laminar boundary layer. In this test problem, it is important that the TKE-ε model be able to reflect the turbulent features of fluid flow at high Reynolds number conditions and that the usage of wall function in the computational domain close to the solid boundaries be valid. The convergence history is shown in Figure 5.21. It is found that the Power-Law scheme shows better convergent characteristics than the QUICK and QUICKM schemes, and the laminar simulation converges better than the turbulent ones for all three methods.
So free boundary $U = 1$

boundary layer

$u = 0, v = 0$ (no-slip condition)

Figure 5.18 Physical domain of the flat-plate flow

Figure 5.19 Numerical domain of the flat-plate flow
Figure 5.20  Velocity profile comparison of the flat-plate flow at $Re=10000$

Figure 5.21  Convergence history of the flat-plate flow at $Re=10000$
5.1.4 2-D Channel Flow

A two-dimensional flow between two parallel plates is the fourth two-dimensional test problem. Again, this simple flow problem, which involves basic boundary conditions encountered in most internal flow problems, is employed for the purpose of studying the laminar and turbulent boundary layers and evaluating the performance of the TKE-\(\epsilon\) turbulence modeling in conjunction with the wall-function method. In this case, a uniform velocity profile enters the channel, and the flow is allowed to fully develop before leaving the channel. Figure 5.22 shows the physical geometry as well as the boundary conditions of this test problem. No-slip conditions are applied to the top and bottom walls, and mass conservation is used to calculate the outlet normal velocity distribution without specifying any outflow boundary conditions. The flow enters the constant area channel and gradually develops along the length of the channel downstream. The channel length is decided according to the channel operation Reynolds number, which is defined by the inlet constant velocity, density, dynamic viscosity and half of the channel height \(h\), i.e., \(\text{Re} = \frac{U \cdot h}{\nu}\), to ensure fully developed flow at the channel outlet. The computational domain consisting of a 102 by 102 stretched grid is shown in Figure 5.23. The cross-stream grid is symmetric about the central line and highly stretched close to the walls in order to resolve the boundary layer. The streamwise grid is uniform. Figure 5.24 shows the horizontal velocity profiles at exit (20 channel heights) using the Power-Law, QUICK and QUICKM schemes for the Reynolds number of 1000. The steep velocity gradient near the wall is clearly captured by the turbulent simulations using all three schemes. For laminar simulations, these three methods produce almost the same results. However, comparing to the data produced by the Power-Law and QUICKM schemes, the QUICK scheme over-predicts the velocity in the central flow region and under-estimates the velocity in the near-wall flow region for turbulent simulation. The convergence performance of these three schemes is shown in Figure 5.25.
$U = 1$

\[ u = 0, \ v = 0 \] (no-slip condition)

boundary layer

hydrodynamic entrance region

fully developed region

Figure 5.22 Physical domain of the channel flow

Figure 5.23 Numerical domain of the channel flow
Figure 5.24  Velocity Profile comparison of the channel flow: $x=20$ channel heights (at exit) and $Re=1000$

Figure 5.25  Convergence history of the channel flow at $Re=1000$
5.1.5 2-D Block Flow

In order to understand the flow phenomena related to the variations of Reynolds numbers and to pave the way for the simulation of a two-dimensional shelterbelt flow, the last test case considered is that of a flow over a two-dimensional square-block mounted on a ground plate. Figure 5.26 shows the physical domain. The upstream boundary of the computational domain is 30 unit-lengths away and the downstream boundary is 50 unit-lengths away. The top boundary is located at a distance of 10 unit-lengths. The solution is solved on a 102 by 46 non-uniform mesh with a highly stretched grid near the body and ground. A partial view of the computational domain is shown in Figure 5.27. A uniform velocity profile is the inlet boundary condition. No-slip conditions are enforced on the body surface and the ground plate. On the top boundary, a freestream boundary condition is applied. During calculation, the velocity and viscosity inside the solid body are reset to zero and infinity, respectively.

To ensure numerical stability and convergent characteristics, the Power-Law scheme is the only method used for this test case. Wall function is applied to the flow regions near the block and the ground plate. Running conditions with Reynolds numbers of 50, 100 and 1000 are used for laminar flow analysis, and turbulent flow simulation starts from Re=1000 and ranges to Re=100000. Three flow recirculation regions, i.e. frontal, top and rear regions, surrounding the square-block are observed. It is found that for both laminar and turbulent simulations, these flow regions continue to extend when the Reynolds number increases. Figure 5.28 illustrates the changing sequences of the streamline contours for all test cases. At a Reynolds number of 1000, both the front and rear recirculation areas from laminar simulation are greater than the predictions from the turbulent simulation. This is because the kinetic energy of fluid particles in turbulent flow is much larger than that of laminar flow. Therefore, the turbulent flow remains attached to the body and the ground much longer than laminar flow.
Figure 5.26 Physical domain of the 2-D block flow

Figure 5.27 Numerical domain of the 2-D block flow
Figure 5.28  Streamline contours of the 2-D block flow for Re = 50, 100, 1000, 10000, and 100000.
5.2 Three-Dimensional Test Cases

5.2.1 3-D Driven-Cavity Flow

A three-dimensional impulsively started lid-driven cavity flow is the first test problem used to verify the three-dimensional codes. The physical domain of the unit cube is shown in Figure 5.29. No-slip conditions are assumed on all solid boundaries. The constant lid velocity, \( U_{lid} \), is set to be unit. No inflow or outflow boundaries need to be specified. Even though the flow is symmetrical about the center plane, the governing equations must be solved for the entire numerical domain to avoid solving the periodic flow at the center plane. A 22 by 22 by 22 non-uniform grid with fine grid toward the boundaries, as shown in Figure 5.30, is employed. The flow is characterized by the Reynolds number, which is defined by:

\[
Re = \frac{U_{lid} h}{\mu_{\infty}}
\]

where \( h \) is the length of the cubic cavity, \( \mu \) is the dynamic viscosity and \( \rho \) is the density of the fluid.

The distribution of the u-velocity at the center plane obtained from the Power-Law, QUICK and QUICKM schemes for laminar simulation are compared with the laminar numerical results from other researcher [75]. Figures 5.31 to 5.33 depict the velocity (u) distributions along the vertical centerline on the xy-center plane. It is evident that QUICK shows overshoots near the bottom boundary for all three test conditions. Since the range of Peclet numbers for the usage of the Power-Law scheme is large for Reynolds numbers of 100 and 400 (64% and 16%), the predictions from QUICKM are almost the same as the ones from Power-Law scheme. At a Reynolds number of 1000, the Peclet number range applicable for using the Power-Law scheme drops to 6%. Therefore, QUICKM begins deviating from the Power-Law scheme. This phenomenon is consistent with the findings for two-dimensional cavity flow. A comparison of convergence history for the Power-Law, QUICK and QUICKM schemes at a Reynolds number of 1000 is given in Figure 5.34. Again, QUICKM offers better convergence characteristics than the QUICK scheme.
Figure 5.29 Physical domain of a three-dimensional cavity flow

Figure 5.30 Grid distribution on the x-y plane of the three-dimensional cavity flow
Figure 5.31  U-velocity profile along the vertical centerline on the $xy$-center plane at $Re=100$ for a three-dimensional cavity flow

Figure 5.32  U-velocity profile along the vertical centerline on the $xy$-center plane at $Re=400$ for a three-dimensional cavity flow
Figure 5.33 U-velocity profile along the vertical centerline on the $xy$-center plane at $Re=1000$ for a three-dimensional cavity flow.

Figure 5.34 Comparison of convergence history at $Re=1000$ for a three-dimensional cavity flow.
5.2.2 3-D Duct Flow

The second test case computed with the QUICK, QUICKM and Power-Law schemes is the three-dimensional square-duct flow. This type of simple, yet important, three-dimensional flow problem often encountered in real-life applications (such as the flow in turbomachinery, the flow over the rear section of a lifting body, and the flow in an opened-channel) is chosen to test the three-dimensional applicability of these numerical schemes. Except for the fact that in nature the square-duct flow is more complicated close to the wall intersections, it has many features in common with the two-dimensional channel flow which has been discussed in section 5.1.4. Figure 5.35 shows the schematic diagram of the square-duct flow field, as well as the boundary conditions used for the numerical simulation. Uniform velocity at the inlet and no-slip conditions at the walls are the boundary conditions. The computational domain is constructed by a 102 by 22 by 22 nonuniform grid as shown in Figure 5.36. Wall function is applied to the flow region near the solid walls. The duct width, $D$, is the characteristic length used to calculate the operational Reynolds number, $Re = \frac{\varepsilon U_{\infty} D}{\nu}$. At a Reynolds number of 1000, the horizontal velocity profiles on the $xy$-centerplane at the station of 30 duct-widths are shown in Figure 5.37. For laminar simulation, all three methods produce almost the same results. However, the QUICKM scheme shows slightly more overshoots at the center than QUICK, which is different from the findings for two-dimensional channel flow. A larger velocity gradient close to the solid boundary is captured for the turbulent simulation.
Figure 5.36  Grid distribution on the x-y plane of the three-dimensional duct flow

Figure 5.37  Horizontal velocity profiles on the xy-centerplane at the station of 30 duct-widths for a fully developed duct flow at Re=1000
6 NUMERICAL SIMULATION OF SHELTERBELT FLOW

6.1 Governing Equations

The Reynolds averaged three-dimensional differential equations governing the turbulent, steady, incompressible shelterbelt flow in Cartesian coordinates take the form:

mass equation:
\[ \frac{\partial(pu)}{\partial x} + \frac{\partial(pv)}{\partial y} + \frac{\partial(pw)}{\partial z} = 0 \]  \hspace{1cm} \text{(6.1)}

u-momentum equation:
\[ \frac{\partial(puu)}{\partial x} + \frac{\partial(puv)}{\partial y} + \frac{\partial(puw)}{\partial z} = \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial u}{\partial z}) - \frac{\partial p}{\partial x} + \left[ \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial x}) \right] - dF_x \]  \hspace{1cm} \text{(6.2)}

v-momentum equation:
\[ \frac{\partial(pvu)}{\partial x} + \frac{\partial(pvv)}{\partial y} + \frac{\partial(pvw)}{\partial z} = \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial v}{\partial z}) - \frac{\partial p}{\partial y} + \left[ \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial y}) \right] - dF_y \]  \hspace{1cm} \text{(6.3)}

w-momentum equation:
\[ \frac{\partial(pwu)}{\partial x} + \frac{\partial(pvw)}{\partial y} + \frac{\partial(pww)}{\partial z} = \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial w}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial w}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial z}) - \frac{\partial p}{\partial z} + \left[ \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial u}{\partial z}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial v}{\partial z}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial w}{\partial z}) \right] - dF_z \]  \hspace{1cm} \text{(6.4)}

TKE equation:
\[ \frac{\partial(pku)}{\partial x} + \frac{\partial(pkv)}{\partial y} + \frac{\partial(pkw)}{\partial z} = \frac{\partial}{\partial x}(\mu_{eff} \frac{\partial k}{\partial x}) + \frac{\partial}{\partial y}(\mu_{eff} \frac{\partial k}{\partial y}) + \frac{\partial}{\partial z}(\mu_{eff} \frac{\partial k}{\partial z}) \]
Dissipation equation:

\[
\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = \frac{\partial (\epsilon \frac{\mu_{\text{eff}}}{\sigma_x} \frac{\partial \epsilon}{\partial x})}{\partial x} + \frac{\partial (\epsilon \frac{\mu_{\text{eff}}}{\sigma_y} \frac{\partial \epsilon}{\partial y})}{\partial y} + \frac{\partial (\epsilon \frac{\mu_{\text{eff}}}{\sigma_z} \frac{\partial \epsilon}{\partial z})}{\partial z} + \frac{\epsilon}{k} (C_1 G - C_2 \rho \epsilon)
\]

where the \(xyz\)-coordinate represents the global coordinate system, and \(dF\)'s, which are functions of the porosity of the shelterbelt, are the drag force terms due to the deficit of momentum when flow moves across the shelterbelt. Detailed discussions of these additional source terms is presented in the following section.

### 6.2 Source Terms due to Drag Forces

Based on fundamental aerodynamics, total drag is composed of normal pressure drag and the skin friction drag. Both drag components are functions of freestream velocity \(V_\infty\), fluid density \(\rho\), reference area \(A\) and the orientation of the object \(\alpha\) (angle of attack). Traditionally, the freestream dynamic pressure, defined as \(\frac{1}{2} \rho \infty V_\infty^2\), is used to normalize all aerodynamic forces acting on solid bodies, such as aircrafts and missiles.

As for the shelterbelt flow, porosity, which determines the structure of shelterbelt, is one of the major factors influencing the shelterbelt flowfield. It is observed that the flowfield near a living-tree shelterbelt is extremely complicated and is a strong function on the permeability of the trees. Therefore, using the local flow conditions to characterize the momentum deficit, which produces the drag terms in the governing equations of the shelterbelt flow, is a proper approach for analyzing the shelter effect of a porous shelterbelt. The drag terms shown in the governing equations are expressed in terms of the local values and local coordinate system. Detailed procedures for transformation between two coordinate systems are given in Appendix E.
If $C_x, C_y, C_z$ are the force coefficients in the global coordinate system, and $C_\theta, C_\phi, C_r$ are the force coefficients in the local coordinate system then:

\[
dF_x = \frac{1}{2} \rho V_{tot}^2 \cdot C_x \cdot dA
\]

\[
= \frac{1}{2} \rho V_{tot} [V_{tot} \cdot \cos \phi \cdot \cos \theta \cdot C_\theta - V_{tot} \cdot \sin \phi \cdot C_\phi + V_{tot} \cdot \cos \phi \cdot \sin \theta \cdot C_r] \cdot dA
\]

\[
= \frac{1}{2} \rho V_{tot} [u \cdot C_r + w \cdot \cos \phi \cdot C_\theta - V_{tot} \cdot \sin \phi \cdot C_\phi] \cdot dA
\]

\[
\Rightarrow dF_x = \frac{1}{2} \rho V_{tot} \cdot dA[u \cdot C_r + w \cdot \cos \phi \cdot C_\theta - V_{tot} \cdot \sin \phi \cdot C_\phi] \quad (6.7)
\]

\[
dF_y = \frac{1}{2} \rho V_{tot}^2 \cdot C_y \cdot dA
\]

\[
= \frac{1}{2} \rho V_{tot} [V_{tot} \cdot \sin \phi \cdot \cos \theta \cdot C_\theta + V_{tot} \cdot \cos \phi \cdot C_\phi + V_{tot} \cdot \sin \phi \cdot \sin \theta \cdot C_r] \cdot dA
\]

\[
= \frac{1}{2} \rho V_{tot} [v \cdot C_r + w \cdot \sin \phi \cdot C_\theta + V_{tot} \cdot \cos \phi \cdot C_\phi] \cdot dA
\]

\[
\Rightarrow dF_y = \frac{1}{2} \rho V_{tot} \cdot dA[v \cdot C_r + w \cdot \sin \phi \cdot C_\theta + V_{tot} \cdot \cos \phi \cdot C_\phi] \quad (6.8)
\]

\[
dF_z = \frac{1}{2} \rho V_{tot}^2 \cdot C_z \cdot dA
\]

\[
= \frac{1}{2} \rho V_{tot} [-V_{tot} \cdot \sin \theta \cdot C_\theta + 0 + V_{tot} \cdot \cos \theta \cdot C_r] \cdot dA
\]

\[
= \frac{1}{2} \rho V_{tot} [w \cdot C_r - V_{xy} \cdot C_\theta] \cdot dA
\]

\[
\Rightarrow dF_z = \frac{1}{2} \rho V_{tot} \cdot dA[w \cdot C_r - V_{xy} \cdot C_\theta] \quad (6.9)
\]

where $V_{tot}$ is the total velocity, which has the velocity components, $V_{xy}$ and $V_{xz}$, on the $x - y$ and $x - z$ planes, respectively.

Special case (1): If $\theta = 90^\circ$, the flow is on the $x - y$ plane, and then, $w = 0$. $C_\theta = 0$, and $V_{tot} = V_{xy}$. Equations 6.7 to 6.9 become:

\[
dF_x = \frac{1}{2} \rho V_{xy} [u \cdot C_r - v \cdot C_\phi] \cdot dA \quad (6.10)
\]

\[
dF_y = \frac{1}{2} \rho V_{xy} [v \cdot C_r + u \cdot C_\phi] \cdot dA \quad (6.11)
\]
\[ dF_z = \frac{1}{2} \rho V_{zy} [0 \cdot C_r - V_{xy} \cdot 0] \cdot dA = 0 \]  

(6.12)

Special case (2): If \( \phi = 0^\circ \), the flow is on the \( x - z \) plane, and then \( v = 0 \). \( C_\phi = 0 \). \( V_{tot} = V_{xz} \), \( V_{tot} \cdot \cos \theta = V_{xz} \cdot \cos \theta = w \). and \( V_{tot} \cdot \sin \theta = V_{xz} \cdot \sin \theta = u \). Equations 6.7 to 6.9 become:

\[ dF_x = \frac{1}{2} \rho V_{xz} [u \cdot C_r + w \cdot C_\theta] \cdot dA \]  

(6.13)

\[ dF_y = \frac{1}{2} \rho V_{xz} [0 \cdot C_r + w \cdot 0 + 0] \cdot dA = 0 \]  

(6.14)

\[ dF_z = \frac{1}{2} \rho V_{xz} [w \cdot C_r - u \cdot C_\theta] \cdot dA \]  

(6.15)

In this study, special case (1) is adapted for the analysis of a two-dimensional shelterbelt. Following the notations and the sign conventions used in aerodynamics, i.e., \( C_r = C_d \), \( C_\phi = -C_f \). Equations 6.10 and 6.11 can be written as:

\[ dF_x = \frac{1}{2} \rho V_{xy} [u \cdot C_d + v \cdot C_f] \cdot dA \]  

(6.16)

and

\[ dF_y = \frac{1}{2} \rho V_{xy} [v \cdot C_d - u \cdot C_f] \cdot dA \]  

(6.17)

where \( C_d \) and \( C_f \) are the normal pressure drag and the skin friction drag, respectively. By using Equations 6.16 and 6.17, the final governing equations for two-dimensional shelterbelt flow become:

mass equation:

\[ \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} = 0 \]  

(6.18)

u-momentum equation:

\[ \frac{\partial (\rho uu)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} = \frac{\partial}{\partial x} (\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{eff} \frac{\partial u}{\partial y}) \]

\[ -\frac{\partial p}{\partial x} + \left[ \frac{\partial}{\partial x} (\mu_{eff} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{eff} \frac{\partial v}{\partial x}) \right] \]

\[ -\frac{1}{2} \rho V_{xy} [u \cdot C_d + v \cdot C_f] \cdot dA \]  

(6.19)
v-momentum equation:
\[
\frac{\partial (\rho vu)}{\partial x} + \frac{\partial (\rho vv)}{\partial y} = \frac{\partial}{\partial x}(\mu_{\text{eff}} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(\mu_{\text{eff}} \frac{\partial v}{\partial y})
\]
\[
-\frac{\partial p}{\partial y} + \left[ \frac{\partial}{\partial x}(\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\mu_{\text{eff}} \frac{\partial v}{\partial y}) \right]
\]
\[
-\frac{1}{2} \rho V_{xy} [v \cdot C_d - u \cdot C_f] \cdot dA
\]
(6.20)

TKE equation:
\[
\frac{\partial (\rho ku)}{\partial x} + \frac{\partial (\rho kv)}{\partial y} = \frac{\partial}{\partial x}(\mu_{\text{eff}} \frac{\partial k}{\partial x}) + \frac{\partial}{\partial y}(\mu_{\text{eff}} \frac{\partial k}{\partial y})
\]
\[
+(G - \rho \varepsilon)
\]
(6.21)

Dissipation equation:
\[
\frac{\partial (\rho \varepsilon u)}{\partial x} + \frac{\partial (\rho \varepsilon v)}{\partial y} = \frac{\partial}{\partial x}(\mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x}) + \frac{\partial}{\partial y}(\mu_{\text{eff}} \frac{\partial \varepsilon}{\partial y})
\]
\[
+ \frac{\varepsilon}{k} (C_1 G - C_2 \rho \varepsilon)
\]
(6.22)

where \( V_{xy} \) denotes the total velocity on the global \( x - y \) plane. Details of linearization of these source terms due to the drag forces are described in Appendix F.

### 6.3 Shelterbelt Simulations

The QUICKM scheme is used to predict the flow pattern for a shelterbelt corresponding to that used in the experiment described by Bradley and Mulhearn [19] and Finnigan and Bradley [20]. The numerical model with 50% porosity is the focus of the numerical investigation in this study. The upstream boundary of the computational domain is 30 shelter-heights away and the downstream boundary is 50 shelter-heights away. The top boundary is located at a distance of 10 shelter-heights. A typical velocity profile in the atmospheric boundary layer, which is consistent with the observed undisturbed velocity profile provided by Wilson [21], is the inlet boundary condition. Again, conservation of
mass to balance the inlet and outlet mass flow rates is used for updating the outflow boundary. The TKE-ε equations are solved in the interior numerical domain and wall function is applied to the flow regions near the solid boundaries. The numerical domain is non-uniformly divided into 30 cells upwind, 68 cells downwind and 2 uniform cells inside the shelterbelt along the streamwise direction. There are 20 uniform cells inside the shelterbelt and 24 non-uniform cells on top of the shelterbelt along the vertical direction. Figures 6.1 and 6.2 show the physical and computational domains, respectively.

Since information about the drag coefficient related to the porosity of the shelterbelt is unavailable, the following approach is employed to investigate the flow field of a shelterbelt with 50% porosity. By neglecting the skin friction drag, a zero porosity (or 100% solidity) shelterbelt is used to estimate the static pressure drop when the flow moves across the shelterbelt. From this simulation a drag coefficient of 1.26 is obtained by normalizing the static pressure drop across the shelterbelt by using the freestream dynamic pressure. It is reasonable to assume that the value of the drag coefficient for a 50% porosity shelterbelt must be between 0 known for a shelterbelt with 100% porosity and the 1.26 calculated for a shelterbelt with 0% porosity. A drag coefficient of 0.15 normalized by the freestream dynamic pressure for a shelterbelt with 50% porosity is obtained by adjusting the value of the drag coefficient until the numerical result matches well with the available observed data.

Figure 6.3 displays the comparisons of vertical profiles of normalized horizontal wind-speed at 4.2 shelter-heights on the leeside for a 50% porosity shelterbelt. The unfilled squares are the freestream velocity distribution from the field observation [21] and the dotted line is the inlet velocity profile used for numerical simulation. The unfilled circles represent the observed data at \( x/H = 4.2 \) on the leeside, where \( H \) denotes the height of shelterbelt. The solid line is the simulated result from Wang and Takle [37]. The numerical data from turbulent modeling using the QUICKM scheme is presented by the dashed line which displays good agreement with the experimental data. It is found that
Figure 6.1  Physical domain of the shelterbelt flow

Figure 6.2  Numerical domain of the shelterbelt flow
a small recirculation region near the shelterbelt on the windside is not observed by the field experiment or in Wang's simulation. The field experiment perhaps did not have any data in this region.

The comparison of horizontal profiles of normalized horizontal windspeed at fixed heights $y/H=0.4$ and $y/H=1.9$ are shown in Figure 6.4. The unfilled circles and triangles stand for the observed data, and the simulated results from Wang are displayed by the solid and dotted curves. In addition, four curves of numerical data using the QUICKM scheme are also plotted. Evidently, the horizontal distribution of the $u$-velocity gradually recovers to the undisturbed condition from $y/H=0.4$, $0.8$, $1.9$ to $4.0$. Both numerical data at $y/H=0.4$ and at $y/H=1.9$ compare well to the observed data. However, the prediction at $y/H=0.4$ gives a smaller rate of recovery towards the upstream conditions. In this study, there is a secondary separation flow occurring at $x/H=25$ downwind, not observed earlier. It is believed that the flow field becomes unsteady when the Reynolds
number is in the range of 1.0E+05 and higher under standard atmospheric conditions, and an average undisturbed windspeed of 3 to 5 meters per second. Therefore, unsteady simulation is required for better predictions of the shelterbelt flow. The vertical $u$-velocity profiles at different leeward locations are depicted in Figure 6.5. A progressive recovery of the $u$-velocity to the freestream condition ($x/H=-15$) is expected in the leeside.

For a better understanding of the relationship between the porosity of the shelterbelt and the drag coefficient normalized by undisturbed dynamic pressure, a series of test conditions with $C_d$ value changing from 0.0 to 1.25 is evaluated. Again, skin friction drag is neglected in this series of tests. Four ranges, namely, an extremely loose range, a loose range, a medium dense range and a dense range, are classified based on the dimensions of the flow recirculation region in the lee of shelterbelt. It is found that no influence on the flowfield for shelterbelts with $C_d$ value less than 0.1.
becomes larger than 0.1. a recirculation region occurs in the lee of shelterbelt. As the value of the drag coefficient increases, the size of the leeward recirculation increases. The recirculation region on the windside also grows. as the $C_d$ increases. At a $C_d$ of 0.27, the leeward recirculation region reaches a maximum with a length of $13 \ H$ and a height of $1.4 \ H$. After that, the length of the recirculation decreases, but the height keeps increasing. As the $C_d$ approaches 0.65, the sizes of the recirculation regions are $2.3 \ H$ in length and $0.6 \ H$ in height for the windward recirculation region. and $7 \ H$ in length and $2.3 \ H$ in height for the leeward one. The dimensions of these two regions remain nearly constant for $C_d$ values larger than 0.65. In other words, shelterbelt porosity higher than the drag coefficient of 0.65 no longer affects the flowfield of shelterbelt.

The results from this investigation, i.e., a medium dense shelterbelt offers the optimal shelter efficiency, is consistent with the findings from Nagaeli, reported by Eimern et al. [6], Alkhallil [4], and Wang and Takle [37]. The relationship between the porosity of the
shelterbelt and the drag coefficient normalized by undisturbed dynamic pressure is illustrated in Figure 6.6. and the streamline plots for various $C_d$ values together with the one for a solid shelterbelt are shown in Figure 6.7.

A second series of analysis is conducted using the drag coefficients normalized by local dynamic pressure, which is denoted by $C_{d_{local}}$, in order to study the complicated flow patterns near or inside the shelterbelt. In this series of test, the skin friction drag coefficient is assumed to be one-tenth of the $C_{d_{local}}$. The $C_{d_{local}}$ is determined by fitting the simulated results to the ones obtained from the first test series under the same running conditions.

The correlation between the drag coefficients normalized by undisturbed and local dynamic pressure are revealed in Figure 6.8. Since the local velocity decreases when the porosity of shelterbelt decreases, the $C_{d_{local}}$ keeps increasing to infinity as the drag coefficient expressed by the undisturbed velocity becomes closer to the upper limit (solid
Figure 6.7 Streamline contours of different drag coefficients normalized by freestream dynamic pressure: $Cd=0.15$, $0.19$, $0.27$, $0.35$, $0.45$, $0.75$, and a solid shelterbelt from bottom to top.
line with $C_{d\text{-solid}} = 1.26$, as shown in Figure 6.8). From the empirical point of view, it is unrealistic to normalize the drag coefficient by local dynamic pressure due to the difficulty of measuring the local velocity for the denser shelterbelt; therefore, a horizontal line drawn on $C_{d\text{-solid}} = 0.3$ indicates that this approach is applicable for simulation of the medium dense shelterbelts.

![Figure 6.8](image.png)

Figure 6.8 Relationship between the drag coefficient normalized by freestream dynamic pressure and the drag coefficient normalized by local dynamic pressure
7 CONCLUSIONS AND RECOMMENDATIONS

The focus of the present study is to develop an unsteady, three-dimensional, incompressible and control volume-based Reynolds averaged Navier-Stokes flow solver to simulate the aerodynamic characteristics of shelterbelt flow.

To develop a better understanding of the characteristics of shelterbelt flow which can help in the numerical modeling, results from two field observations were used. These provided the primary air field information of living-tree shelterbelts under atmospheric boundary conditions. A wind-tunnel flow visualization of scaled-model fences was also conducted for exploring the fundamental phenomena of the flow patterns surrounding the shelterbelts. It is found that: 1) The incident angle of the approaching flow is strongly related to the leeward windspeed reduction close to the ground (less than 0.5 shelter-heights). A large angle of incidence causes an increase in the distance travelled by the flow through the shelterbelt, which results in the skin friction force between the air and the shelter elements increasing; therefore, a significant windspeed reduction occurs on the leeside. 2) The static pressure gradient between the windside and leeside of the shelterbelt increases when the approaching windspeed increases. 3) For both two-dimensional and three-dimensional models under the same flow conditions, the wake area decreases when the porosity of the model fence increases. 4) Increasing the incident angle results in a decrease of the wake area, which contradicts the findings described in 1) and 2). This situation happens because the thickness of the model is so minimal that the friction effect induced by the airflow penetrating the fence is less significant than that of a living-tree shelterbelt. Furthermore, when the model becomes more parallel
to the approaching flow, the form drag decreases due to a smaller reference area (or so
called wetted area): therefore, a smaller wake area behind the fence is expected. 5) If two
comparable models have the same porosity and same orientation, the wake area in the
leeward of a two-dimensional fence is always less than the one behind a three-dimensional
fence for all test cases.

The Power-Law scheme fails to reflect the turbulence characteristics of shelterbelt
flow by dropping the diffusion terms when the absolute value of the Peclet number is
greater than ten. A new formulation to represent the interface convective kinematics
which uses the Lagrange interpolation to correlate the primary variables ($u$ and $v$) among
the main grid and the upstream and downstream nodal values has been developed to
remedy this problem. This technique, which automatically satisfies the Scarborough
criterion, allows the discretized governing equation to be formulated as a combination
of first-order terms and source terms in an efficient and natural manner without loss
of generality. Furthermore, two higher-order schemes, SOU and QUICK, are combined
with the Power-Law scheme in order to retain accuracy without losing stability and
convergence characteristics.

A two-dimensional driven-cavity flow problem is used to assess and verify the perfor­
mane of these newly developed schemes, namely, SOU, SOUM, QUICK and QUICKM.
The conclusions are summarized as follows: 1) The SOU and QUICK schemes with
higher-order accuracy features are certainly better models than the Power-Law scheme
for investigating fluid flow problems with higher Reynolds number flow conditions. How­
ever, these higher-order schemes not only suffer from poor convergence, but also overesti­
mate as the Reynolds number increases. A cautious adjustment of the relaxation factors
and the correct choice of grid in the computational domain are necessary for ensuring the
stability and obtaining convergent solutions. 2) The Power-Law scheme is accurate for
simulating flow problems with lower Reynolds numbers. Excellent stability and conver­
gence characteristics observed for all the cases in this study reveal the robustness of the
Power-Law scheme. 3) The SOUM and QUICKM modified higher-order schemes are formulated in an attempt to combine the advantages of both the Power-Law scheme and the higher-order schemes while avoiding the drawbacks of both. Performance of these modified schemes is shown at a Reynolds number of 400 with 22 by 22 and 42 by 42 grid spacings for the driven-cavity problem. The advantages of good convergence characteristics start to diminish as the Reynolds number increases or the resolution of the computational domain becomes higher. This is due to the fact that the range of Peclet numbers becomes larger when the Reynolds number increases: therefore, the range of using the Power-Law scheme decreases. 4) Further study should be focused on the range for implementing the Power-Law scheme with a combination of higher-order schemes in order to fully understand the characteristics of these modified schemes.

In this study, the TKE-ε model is adopted for simulation of turbulent flow, the characteristic of shelterbelt flow under the atmospheric conditions. The TKE-ε model is applicable in a fully turbulent flow. However, close to solid walls and some other interfaces, there are inevitably regions where the local Reynolds number of turbulence is so small that molecular viscous effects dominate the turbulent effects. The molecular viscosity may also be of the same order or larger than the turbulent viscosity, rendering the TKE-ε invalid. The wall-function method was employed to solve this problem for reasons of CPU time and storage.

After using seven classical test problems (five 2-D and two 3-D test cases) to check the performance of the Power-Law, QUICK and QUICKM schemes, the QUICKM scheme and the TKE-ε model for the general domain, and the wall function for regions close to the solid boundaries were found to be suitable for simulating the shelterbelt flow. For better predictions, drag terms due to pressure and skin friction, which all artificial windbreaks and living-tree shelterbelts possess, have been included in the momentum equations. The drag coefficients obtained by normalizing the static pressure drop (using either freestream or local dynamic pressure) are analyzed to establish the relationship
with the porosity of the shelterbelt.

A numerical model with 50% porosity is the focus of this series of numerical investigations. First, by neglecting the skin friction drag, a value for the normal pressure drag coefficient of 1.26 for a solid shelterbelt model is obtained by normalizing the static pressure drop across the shelterbelt using freestream dynamic pressure. A drag coefficient of 0.15, normalized by freestream dynamic pressure for a shelterbelt with 50% porosity, is obtained by adjusting the value of $C_d$ until the numerical result matched well with the available observed data. The simulated results agree well with the experimental data.

In order to fully understand the relationship between the porosity of the shelterbelt and the drag coefficient normalized by the undisturbed dynamic pressure, the first series of investigations with $C_d$ value changing from 0.0 to 1.25 is implemented after neglecting the skin friction drag. Four distinct zones are characterized based on the size of the flow separation region in the lee of the shelterbelt. The simulated results reveal that the maximum protected distance in the leewside occurs for a shelterbelt with a porosity of 20% to 40%. This conclusion is consistent with the findings from field observations and numerical simulations by other researchers.

Assuming that the skin friction drag coefficient is one-tenth of the $C_{d_{\text{local}}}$, the correlation between the drag coefficients normalized by either undisturbed or local dynamic pressure is studied in a second series of analyses. Since the local velocity becomes zero at the stagnation point on the solid model, the $C_{d_{\text{local}}}$ approaches infinity; therefore, using the drag coefficient normalized by the local dynamic pressure is limited to the simulation of the flow field of a shelterbelt with higher porosity. However, this approach is valid and proper for analyzing medium-dense shelterbelts, which are used very often in real-life application. The overall simulation of the shelterbelt flow using this newly developed numerical scheme shows satisfactory agreement with both field experiments and other available numerical model results. Therefore, it has proved to be a useful tool for performing further analysis of shelterbelt aerodynamics.
The recommendations for possible areas of future research and improvements to the current study are highlighted next:

1) The shelterbelt flow in the atmospheric boundary layer has to be studied as an unsteady flow.

2) The energy equation associated with conservation of heat needs to be added to the governing equations in order to include the thermal effects due to temperature stratification in the atmospheric boundary layer.

3) A series of wind tunnel investigations is required to provide proper information for building a relationship between the drag coefficient and the porosity of the model-shelterbelt.

4) The relationship of the geometric characteristics of the shelterbelt, such as the height, the width, the aspect ratio (ratio of the shelter-span to the shelter height), and the shelter efficiency is a very important topic for further investigation.

5) It is observed that the wind in a neutral environment rarely blows perpendicular to a shelterbelt. Oblique wind significantly affects the windspeed reduction on the leeside of a shelterbelt. Therefore, a thorough study of the relation between the incident angle of the approaching wind and the shelter effect is required.

6) The numerical model developed in this current study is capable of simulating a shelterbelt with a non-uniform distribution of porosity. This technique, which uses the drag coefficient normalized by local dynamic pressure to characterize the momentum deficit, offers significant capabilities for more complicated shelterbelt configurations as well.
APPENDIX A  ZERO-EQUATION TURBULENCE MODELS

Two famous algebraic turbulence models, namely the Cebeci-Smith model [76] and the Baldwin-Lomax model [77] were tested along with TKE-\(\epsilon\) model during the turbulent model selection process and are presented here for completion.

Cebeci-Smith Model

The turbulent boundary layer is considered to be formed by two regions, an inner region and an outer region, with different expressions for the eddy viscosity coefficient due to the turbulent intensity generated by the solid boundary.

The shearing stress on the wall is defined as:

\[ \tau_w = \mu_l \left( \frac{\partial u}{\partial y} \right) \]  \hspace{1cm} (A.1)

and the friction velocity is defined by:

\[ u_r = \sqrt{\frac{\tau_w}{\rho}} \]  \hspace{1cm} (A.2)

The effective viscosity is the sum of the laminar molecular viscosity, which is a constant, and the turbulent dynamic viscosity, i.e.,

\[ \mu_{eff} = \mu_l + \mu_t \]  \hspace{1cm} (A.3)

The vorticity is defined as:

\[ | \vec{\omega} | = | \nabla \times \vec{V} | \]
For two-dimensional flow:

\[ | \vec{w} | = \sqrt{\left( \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2} \]  \hspace{1cm} (A.4)

For three-dimensional flow:

\[ | \vec{w} | = \sqrt{\left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right)^2 + \left( \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right)^2} \]  \hspace{1cm} (A.5)

The value of parameter, \( A^+ \), is either 26 or is given by:

\[ A^+ = 26 \left[ 1 + \frac{y \left( \frac{\partial v}{\partial x} \right)}{\rho u_r^2} \right]^{\frac{1}{2}} \]  \hspace{1cm} (A.6)

The variable length, \( y^+ \), is defined by:

\[ y^+ = \frac{y u_r}{v_l} \]  \hspace{1cm} (A.7)

For the inner region, the turbulent viscosity can be calculated using the following formula:

\[ \mu_{t, inner} = \rho \cdot l_{mix}^2 \cdot | \vec{w} | \]  \hspace{1cm} (A.8)

where \( l_{mix} \) is defined by:

\[ l_{mix} = \kappa \cdot y \cdot \left[ 1 - e^{\left( \frac{y^+}{\kappa} \right)} \right] \]  \hspace{1cm} (A.9)

and \( \kappa = 0.41 \) is the Von Karman constant while \( y \) is the geometric coordinate.

For the outer region, the boundary displacement thickness is defined as:

\[ \delta^* = \int_0^\delta \left[ 1 - \frac{u(y)}{U_e} \right] dy \]  \hspace{1cm} (A.10)

where \( U_e \) is the external velocity at the boundary layer edge.

The function \( F \) represents the influence of the intermittency at the edge of the boundary layer and is given by the empirical formula:

\[ F = \frac{1}{1 + 3.5\left( \frac{y}{\delta} \right)} \]  \hspace{1cm} (A.11)
where $\delta$ is the boundary layer thickness. Therefore, the turbulent viscosity can be found using the following equation:

$$\mu_{t,outer} = \alpha \cdot \rho \cdot U_e \cdot \delta^* \cdot F \quad (A.12)$$

The switch from the inner to the outer value of the turbulent viscosity occurs at the position $y_c$, where the inner value becomes equal to the outer value as shown in Figure A. that is:

$$\mu_t = \mu_{t,outer} \quad if \quad y > y_c$$

$$\mu_t = \mu_{t,inner} \quad if \quad y < y_c$$

The main disadvantage of using the Cebeci-Smith model is the difficulty of calculating the velocity at the edge of the boundary layer and the local boundary thickness. Therefore, the evaluation of the boundary displacement thickness shown in Equation A.10 is nearly impossible. In order to remedy this problem, Baldwin and Lomax modified the estimation of the outer part of the eddy viscosity. This modification is shown in the following section.

**Baldwin-Lomax Model**

The definitions of these parameters are exactly the same as those defined in the Cebeci-Smith model.

For the inner region:

$$\mu_{t,inner} = \rho \cdot l_{mix}^2 \cdot |\vec{w}| \quad (A.13)$$

For the outer region:

$$\mu_{t,outer} = 0.0168 \cdot \beta \cdot F \cdot y_{max} \cdot \Gamma_{max} \quad (A.14)$$

where the intermittency function $F$ is expressed by:

$$F = \frac{1}{1 + 5.5 \left( \frac{\sigma_y}{y_{max}} \right)^6} \quad (A.15)$$
Figure A.1  Two-layer edge viscosity model
and the parameters $\alpha$ and $\beta$ are 0.3 and 1.6, respectively.

The function $\Gamma$ is defined as:

$$\Gamma = y[1 - e^{\left(\frac{-y}{\alpha + 1}\right)}] \cdot |\bar{u}|$$  \hspace{1cm} (A.16)

and $y_{max}$ is the maximum value of $y$ when $\Gamma$ attains its maximum value, $\Gamma_{max}$. 
APPENDIX B  FORMULATION OF HIGHER-ORDER ACCURACY SCHEMES

Governing Equations

The governing equations for the two-dimensional steady convection-diffusion transport of \( \phi \), any dependent variable, written in a Cartesian coordinate system is:

\[
\frac{\partial (\rho \phi)}{\partial x} + \frac{\partial (\rho \phi)}{\partial y} = \frac{\partial}{\partial x} \left( \Gamma_{\phi} \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma_{\phi} \frac{\partial \phi}{\partial y} \right) + S_{\phi} \quad (B.1)
\]

where \( \Gamma_{\phi} \) is the diffusivity and \( S_{\phi} \) is the source.

The discretized form of the above generic equation integrated over a typical control volume is given by:

\[
ap \phi_p = a_E \phi_E + a_W \phi_W + a_N \phi_N + a_S \phi_S + b 
\quad (B.2)
\]

where \( a's \) are coefficients made up of diffusive and convective fluxes and \( b \) is the source term. Adopting Patankar's notation [51], the coefficients \( a_E, a_W, a_N, a_S \) and \( a_P \) of the above equation for various schemes are compactly given by:

\[
a_E = D_{\phi} A(|P_r|) + [-F_{\phi}.0] : \quad a_W = D_{\phi} A(|P_w|) + [F_{\phi}.0] \\
a_N = D_{\phi} A(|P_n|) + [-F_{\phi}.0] : \quad a_S = D_{\phi} A(|P_s|) + [F_{\phi}.0] \\
a_P = a_E + a_W + a_N + a_S
\]

where \( D \) is the diffusional flux coefficient, \( F \) is the mass flux through the control surface and \( P \) is the grid Reynolds number (Peclet number) given by \( P = F/D \). \( A(|P|) \) is
a function of the local Peclet number and takes on different values depending on the schemes being used. For the first-order upwinding scheme, \( A(|P|) \) takes the value of 1, and for Power-Law scheme, \( A(|P|) = 0.1 - 0.1|P| \).

Leaving the diffusive terms unchanged, if we focus on the integration of the convection terms of the conservation equation, we obtain:

\[
F_e \phi_e - F_w \phi_w
\]  

(B.3)

to be the algebraic form of flux balance of \( \phi \) across a one-dimensional control volume. Henceforth, in the interest of brevity, we shall consider only this term. Two-dimensional and three-dimensional situations are straightforward extensions of the concept to be outlined and will be left out of the main discussion.

It is the stencil used in the estimation of the facial values \( \phi_w \) and \( \phi_e \) that determines the order of accuracy of the convective transport of \( \phi \). For example, the first-order upwinding convective flux \( F_w \phi_w \) at the interface \( w \) can be algebraically stated using Figure B.1 as:

\[
F_w \phi_w = [F_w.0] \phi_w - [-F_w.0] \phi_p
\]  

(B.4)

The above expression literally translates to \( \phi_w \) being assigned \( \phi_w \) when \( F_w \) is positive and \( \phi_p \) when \( F_w \) is negative, which is an upstream biased, piecewise constant profile.

![Figure B.1 One-dimensional grid format](image-url)
of $\phi$: when the estimation of $\phi$ at the interface is to be done with anything but a piecewise constant profile of $\phi$, the interpolation stencils become complicated and the resulting algebraic equations lack generality and require careful manipulations so as not to violate the Scarborough criterion. It is here that we find the Lagrange interpolation and its properties very useful. Lagrange interpolation is very general in that any number of neighbors can be considered for the interpolation of the control volume face value with a prescribed upwind bias. Further, the properties of Lagrange interpolation lend themselves in the manipulation of the algebraic equations to special forms required for the strict adherence of the Scarborough criterion. The use of Lagrange interpolation in modeling convective kinematics with the help of two popular upwinding biased higher-order schemes, namely, second-order upwind (SOU) and QUICK, is illustrated, and the results from using SOUM and QUICKM schemes to simulate the two-dimensional driven-cavity flow problem are presented as well.

Second-Order Upwind Scheme

This scheme estimates the interface value by extrapolating from the two upwind neighbors of the control volume face. Using Lagrange interpolation, the interface value at $w$ can be written as:

For $F_w > 0$:

$$\phi_w = \left( \frac{x_w - x_{WW}}{x_W - x_{WW}} \right) \phi_W + \left( \frac{x_w - x_W}{x_{WW} - x_W} \right) \phi_{WW}$$

or

$$\phi_w = \alpha'_w \phi_W + \alpha_w \phi_{WW}$$

This equation needs to be manipulated in such a way that it facilitates the casting of the final algebraic equation as a first-order upwind term plus additional source terms. Further, it seeks avenues to linearize the source terms to improve the stability of the numerical scheme. It is here that the property of the coefficients of the Lagrange
interpolation becomes particularly useful. By definition, the coefficients of the Lagrange interpolation sum to unity. For the second-order upwind scheme:

\[ \alpha'_w + \alpha_w = 1 \Rightarrow \alpha'_w = 1 - \alpha_w \]  

(B.5)

With the help of the sum of the coefficients property, the interface value is written as:

\[ \phi_w = (1 - \alpha_w) \phi_W + \alpha_w \phi_{WW} \]  

(B.6)

or

\[ \phi_w = \phi_W + \alpha_w (\phi_{WW} - \phi_W) \quad as \quad F_w > 0 \]  

(B.7)

Similarly, for \( F_w < 0 \), the interface value is:

\[ \phi_w = \phi_P + \beta_w (\phi_E - \phi_P) \]  

(B.8)

where

\[ \beta_w = \frac{x_w - x_P}{x_E - x_P} \]  

(B.9)

Combining both of these conditions, one can write the convective flux at the west interface as:

\[ F_w \phi_w = [F_w, 0] [\phi_W + \alpha_w (\phi_{WW} - \phi_W)] - [-F_w, 0] [\phi_P + \beta_w (\phi_E - \phi_P)] \]  

(B.10)

Rewriting this equation with first-order upwind terms as the basis, we get:

\[ F_w \phi_w = [F_w, 0] \phi_W - [-F_w, 0] \phi_P + \{[F_w, 0] \alpha_w (\phi_{WW} - \phi_W) - [-F_w, 0] \beta_w (\phi_E - \phi_P)\} \]  

(B.11)

Similarly, the convective flux on the east face can be written as:

\[ F_e \phi_e = [F_e, 0] \phi_P - [-F_e, 0] \phi_E + \{[F_e, 0] \alpha_e (\phi_{WW} - \phi_P) - [-F_e, 0] \beta_e (\phi_E - \phi_P)\} \]  

(B.12)

The balance of convective flux across the control volume then becomes:

\[ F_e \phi_e - F_w \phi_w = [F_e, 0] \phi_P - [-F_e, 0] \phi_E - [F_w, 0] \phi_W + [-F_w, 0] \phi_P \]
\begin{align}
&+\{[F_e.0] \alpha_e(\phi_W - \phi_P) - [-F_e.0] \beta_e(\phi_{EE} - \phi_E)\} \\
&-\{[F_w.0] \alpha_w(\phi_{WW} - \phi_W) - [-F_w.0] \beta_w(\phi_E - \phi_P)\} \\
\end{align}
(B.13)

In view of the geometric property that \(\alpha_w\), \(\beta_w\), \(\alpha_e\) and \(\beta_e\) are always negative, the algebraic discretized equation for the transport of \(\phi\) across the control volume using second-order upwind for the convective terms and central difference for the diffusion terms is written as:

\[ a_P\phi_P = a_E\phi_E + a_W\phi_W + b \]  
(B.14)

where

\[ a_E = D_e + [-F_e.0] \]  
(B.15)

\[ a_W = D_w + [F_w.0] \]  
(B.16)

\[ a_P = a_W + a_E - \{\alpha_e[F_e.0] + \beta_w[-F_w.0]\} \]  
(B.17)

and

\[ b = \{[-F_e.0] \beta_e(\phi_{EE} - \phi_E) - [F_e.0] \alpha_e\phi_W \}
-\{[F_w.0] \alpha_w(\phi_{WW} - \phi_W) - [-F_w.0] \beta_w\phi_E\} \]  
(B.18)

\[ +[F_w.0] \alpha_w(\phi_{WW} - \phi_W) - [-F_w.0] \beta_w\phi_E \]  
(B.19)

**QUICK Scheme**

As a second illustration of the technique, we consider the popular QUICK scheme developed by Leonard [66]. The interface value is interpolated with two upwind biased neighbors of the interface and one downstream of the interface. Assuming that the values at these grid points are known, Lagrange interpolation yields the interface value at the west face as:

For \(F_w > 0\):

\[ \phi_w = \gamma_{2w}\phi_P + \gamma_w'\phi_W + \gamma_{1w}\phi_{WW} \]  
(B.20)
where
\[ \gamma_w = \frac{X_w - X_p}{X_p - X_w} \left( \frac{X_w - X_{WW}}{X_p - X_{WW}} \right) \]
\[ \gamma'_w = \frac{X_w - X_p}{X_p - X_w} \left( \frac{X_w - X_{WW}}{X_{WW} - X_w} \right) \]
\[ \gamma_{1w} = \frac{X_w - X_p}{X_{WW} - X_p} \left( \frac{X_w - X_{WW}}{X_{WW} - X_w} \right) \]  \hspace{1cm} (B.21)

Upon using the sum of the coefficients rule:
\[ \gamma'_w = 1 - (\gamma_{1w} + \gamma_w) \]  \hspace{1cm} (B.22)

the facial value becomes:
\[ \phi_w = \gamma_{2w} \phi_p + \{ 1 - (\gamma_{1w} + \gamma_w) \} \phi_w + \gamma_{1w} \phi_{WW} \]  \hspace{1cm} (B.23)

A simple rearrangement offers:
\[ \phi_w = \phi_w + \{ \gamma_{2w} \phi_p + \gamma_{1w} \phi_{WW} - (\gamma_{1w} + \gamma_w) \phi_w \} \]  \hspace{1cm} (B.24)

Similarly, for \( F_w < 0 \), we get:
\[ \phi_w = \phi_p + \{ \delta_{1w} \phi_W + \delta_{2w} \phi_E - (\delta_{1w} + \delta_{2w}) \phi_p \} \]  \hspace{1cm} (B.25)

where
\[ \delta_{1w} = \frac{X_w - X_p}{X_p - X_w} \left( \frac{X_w - X_E}{X_w - X_E} \right) \]
\[ \delta'_w = \frac{X_w - X_p}{X_p - X_w} \left( \frac{X_w - X_E}{X_p - X_E} \right) \]
\[ \delta_{2w} = \frac{X_w - X_p}{X_E - X_p} \left( \frac{X_w - X_E}{X_E - X_p} \right) \]  \hspace{1cm} (B.26)

Combining all the conditions for \( F_w \), the flux at the west face can be written as:
\[ F_w \phi_w = \left[ F_w, 0 \right] \phi_w - \left[ -F_w, 0 \right] \phi_p + \left\{ [\gamma_{1w} \phi_W + \gamma_{2w} \phi_W - (\gamma_{1w} + \gamma_w) \phi_W][F_w, 0] \right\} \]
\[ -[\delta_{1w} \phi_W + \delta_{2w} \phi_E - (\delta_{1w} + \delta_{2w}) \phi_p][-F_w, 0] \} \]  \hspace{1cm} (B.27)

The convective flux at the east face of the control volume can be conveniently written as:
\[ F_e \phi_e = \left[ F_e, 0 \right] \phi_p - \left[ -F_e, 0 \right] \phi_E + \left\{ [\gamma_{1e} \phi_W + \gamma_{2e} \phi_E - (\gamma_{1e} + \gamma_{2e}) \phi_p][F_e, 0] \right\} \]
\[
-\{\delta_{1e}\phi_P + \delta_{2e}\phi_{EE} - (\delta_{1e} + \delta_{2e})\phi_E\}[-F_e.0]\} \quad (B.28)
\]

The one-dimensional convective flux conservation across the control volume surrounding \(P\), the main grid point, is therefore given as:

\[
F_e\phi_e - F_w\phi_w = [F_e.0]\|\phi_P - [-F_e.0]\|\phi_E - [F_w.0]\|\phi_W + [-F_w.0]\|\phi_P
\]

\[
+\{\gamma_{1e}\phi_W + \gamma_{2e}\phi_E - (\gamma_{1e} + \gamma_{2e})\phi_P\}[F_e.0]
\]

\[
-\{\delta_{1e}\phi_P + \delta_{2e}\phi_{EE} - (\delta_{1e} + \delta_{2e})\phi_E\}[-F_e.0]
\]

\[
-\{\gamma_{1w}\phi_{WW} + \gamma_{2w}\phi_P - (\gamma_{1w} + \gamma_{2w})\phi_W\}[F_w.0]
\]

\[
+\{\delta_{1w}\phi_W + \delta_{2w}\phi_E - (\delta_{1w} + \delta_{2w})\phi_P\}[-F_w.0]
\] \quad (B.29)

Considering that the geometric properties \(\delta_{2w}\) and \(\gamma_{1e}\) are always less than zero, and treating the diffusive flux across the control volume using the central difference stencil, the discretized formulation for the convection-diffusion equation in the QUICK scheme becomes:

\[
ap\phi_P = a_E\phi_E + a_W\phi_W + b
\] \quad (B.30)

where

\[
a_E = D_e + [-F_e.0] \quad (B.31)
\]

\[
a_W = D_w + [F_w.0] \quad (B.32)
\]

\[
ap = a_W + a_E - (\gamma_{1e}[-F_e.0] + \delta_{2w}[-F_w.0]) \quad (B.33)
\]

and

\[
b = \{\delta_{1e}\phi_P + \delta_{2e}\phi_{EE} - (\delta_{1e} + \delta_{2e})\phi_E\}[-F_e.0]
\]

\[
-\{\gamma_{1e}\phi_W + \gamma_{2e}(\phi_E - \phi_P)\}[F_e.0]
\]

\[
+\{\gamma_{1w}\phi_{WW} + \gamma_{2w}\phi_P - (\gamma_{1w} + \gamma_{2w})\phi_W\}[F_w.0]
\]

\[
-\{\delta_{1w}(\phi_W - \phi_P) + \delta_{2w}\phi_E\}[-F_w.0]
\] \quad (B.34)
Test Cases

A two-dimensional circulating flow of a fluid in a square cavity with three bounded sides and the top wall moving with constant speed is the test problem chosen to assess the performance of using SOU, QUICK, SOUM and QUICKM schemes for modeling the convective kinematics of the convection-diffusion equation. Three cases with Reynolds numbers ranging from 400 to 3200 and three stretched grid formations are studied, namely, 22 by 22, 42 by 42 and 82 by 82. The results of some cases have been shown in Chapter 5.

From the figures showing the horizontal velocity distributions along the vertical centerlines for all test cases, it is evident that the results from the Power-Law scheme always show underestimation with respect to the reference solution, and the deviation becomes larger as Reynolds number increases. For low Reynolds number cases, SOU and QUICK schemes can accurately model the centerline velocity distribution; however, for higher Reynolds number cases, the results obtained by using these higher-order schemes display significant overshoots compared to the benchmark solution. In contrast, SOUM and QUICKM schemes show better agreement with the reference solution, especially when the Reynolds number is 3200 for all the three grids chosen.

As the grid is refined in the computational domain, the differences between the solutions of the SOU, QUICK, SOUM and QUICKM schemes become insignificant. The Power-Law method is seen to be clearly the least accurate for increasing Reynolds number flows. SOUM and QUICKM schemes always fall between the Power-Law scheme and the higher-order schemes. It is observed that SOUM and QUICKM schemes start deviating from the Power-Law scheme at Re=400 and begin approaching the SOU and QUICK schemes at higher Reynolds number cases. This is because the range of Peclet number for the low Reynolds number case is much smaller than that of the high Reynolds number case: the region eligible for using the Power-Law scheme in the numerical domain
becomes smaller as the Reynolds number increases.

From the plots of convergence history (overall residual versus iteration number) for all test cases, it is observed that the Power-Law scheme preserves better computational stability and displays excellence convergence characteristics. SOU and QUICK schemes do not behave as well as the Power-Law scheme in terms of the convergence characteristic which is, in general, the known property of the higher-order schemes.

In most cases, QUICK shows better convergence features than the SOU scheme. For lower Reynolds numbers and coarse grid cases, the SOUM and QUICKM schemes exhibit the best convergence history compared to the SOU and QUICK schemes, especially, for a Reynolds number of 400 for the 22 by 22 grid case. This benefit diminishes as the Reynolds number becomes higher, both for coarse and fine grids. Again, this is due to the reduced involvement of the Power-Law scheme when implementing these combination schemes as the cell Reynolds number increases. It is observed that SOUM and QUICKM schemes only offer better stability characteristics within a certain percentage of the range of Peclet numbers in which the Power-Law scheme is implemented. From Figure 5.5 (shown in Chapter 5), the Peclet number ranges from -16 to 37 and 23% of the time the Power-Law scheme is used. SOUM shows excellent convergence behavior. From Figure B.10, the portion using Power-Law scheme becomes 43%, and the SOUM scheme only gives an overall residual of four or five orders of magnitude before it stalls. Figure B.13 shows a portion of 86% using the Power-Law scheme; therefore, the convergent history for both the SOUM and QUICKM schemes is almost the same as that of the Power-Law scheme. It appears that use of the Power-Law scheme mixed with higher-order schemes should be either weak enough not to interfere with the original higher-order scheme or strong enough to ensure numerical stability, and it seems that this crucial Peclet number range for using the Power-Law scheme is a problem-dependent criterion. Fortunately, for the lower Reynolds number and coarse grid cases, the SOUM and QUICKM schemes are capable of providing accurate predictions and display better stability characteristics.
Figure B.2 U-velocity profile on the vertical centerline of the driven-cavity at $Re=3200$ with a 22 by 22 non-uniform grid.

Figure B.3 U-velocity profile on the vertical centerline of the driven-cavity at $Re=400$ with a 42 by 42 non-uniform grid.
Figure B.4  U-velocity profile on the vertical centerline of the driven-cavity at Re=1000 with a 42 by 42 non-uniform grid

Figure B.5  U-velocity profile on the vertical centerline of the driven-cavity at Re=3200 with a 42 by 42 non-uniform grid
Figure B.6  U-velocity profile on the vertical centerline of the driven-cavity at Re=400 with a 82 by 82 non-uniform grid

Figure B.7  U-velocity profile on the vertical centerline of the driven-cavity at Re=1000 with a 82 by 82 non-uniform grid
Figure B.8  U-velocity profile on the vertical centerline of the driven-cavity at Re=3200 with a 82 by 82 non-uniform grid

Figure B.9  Convergence history at Re=3200 with a 22 by 22 non-uniform grid for the driven-cavity flow
Figure B.10  Convergence history at Re=400 with a 42 by 42 non-uniform grid for the driven-cavity flow

Figure B.11  Convergence history at Re=1000 with a 42 by 42 non-uniform grid for the driven-cavity flow
Figure B.12 Convergence history at $Re=3200$ with a 42 by 42 non-uniform grid for the driven-cavity flow

Figure B.13 Convergence history at $Re=400$ with a 82 by 82 non-uniform grid for the driven-cavity flow
Figure B.14  Convergence history at Re=1000 with a 82 by 82 non-uniform grid for the driven-cavity flow

Figure B.15  Convergence history at Re=3200 with a 82 by 82 non-uniform grid for the driven-cavity flow
APPENDIX C ROUGHNESS-SPIRE DESIGN

Spire and roughness combinations are designed to provide a turbulent boundary of a certain thickness at a distance equal to six times the spire height downstream [78]. With this design, it is possible to build up the appropriate wind tunnel configuration to simulate the atmospheric boundary layer characteristic of the flow past scaled shelterbelts. The following procedures describe the details used in generating a required turbulent boundary layer thickness occurring at the station where the model shelterbelts are located in the test section of the environmental wind tunnel at the Iowa State University AEEM Department. Basic conditions are given as: the height of the test section of the environmental wind tunnel at Iowa State University, $H_0$, is 122 centimeters. The boundary layer thickness, $\delta_{6H}$, at a distance six times the spire height downwind, is 20 centimeters. The skin friction coefficient of the spires, $C_f$, is 0.0032, and the distance between the roughness blocks, $D$, is 15 centimeters. The spire spacing is half of the spire height. It is necessary to find the dimensions and spacings of the spires, as well as the roughness blocks, so that the required thickness of the boundary layer at the location where the model shelterbelt is placed can be estimated.

The roughness-spire design is based on a power law wind profile assumption at the station six times spire height downwind, i.e.,

$$\frac{u}{u_\delta} = \left(\frac{Z}{\delta}\right)^{0.4}$$

(C.1)

where $u_\delta$ is the undisturbed velocity, which is approximately equal to the inlet velocity
of the wind tunnel, and \( u \) is the velocity at a distance \( Z \) from the wall in the boundary layer (see Figure C.1). The value of \( \alpha \) depends on the desired value of the friction coefficient, \( C_f \), which is given as:

\[
C_f = 0.136\left(\frac{\alpha}{1 + \alpha}\right)^2 \tag{C.2}
\]

The following empirical equations are used to calculate the parameters needed in the design process:

\[
\beta = \left(\frac{\delta}{H_o}\right)(\frac{\alpha}{1 + \alpha}) \tag{C.3}
\]

where \( H_o \) is the test section height. The spire height \( H \) is given as:

\[
H = 1.39\left(\frac{\delta}{1 + \frac{\alpha}{2}}\right) \tag{C.4}
\]

and

\[
\Psi = 3\left\{\frac{2}{1 + 2\alpha} + \beta - \frac{1.13\alpha}{(1 + \alpha)(1 + \frac{\alpha}{2})}\right\} \frac{1}{(1 - \beta)^2} \tag{C.5}
\]

\[
\frac{b}{H} = 0.5H_o\Psi\left\{\frac{1 + \frac{\alpha}{2}}{(1 + \Psi)^\delta}\right\} \tag{C.6}
\]

where \( b \) is the spire width. The pressure drop factor \( F \) is given as:

\[
F = \left\{1 + \frac{\delta}{H_o}\left[\frac{\alpha(3 + 2\alpha)}{1 + \alpha(1 - \frac{\alpha}{H_o})}\right]\right\}^{-1} \tag{C.7}
\]

The increment of \( \delta \) with \( \Delta x \) downwind of six spire heights is:

\[
\Delta\delta = 0.068\alpha\left\{\frac{1 + 2\alpha}{1 + \alpha}\right\}\Delta x F \tag{C.8}
\]

The roughness height \( k \) is defined as:

\[
k = \delta \exp\left\{(\frac{2}{3})\ln\left(\frac{D}{\alpha}\right) - 0.1161\sqrt{\frac{2}{C_f}} + 2.05\right\} \tag{C.9}
\]

where \( D \) is the spacing of the roughness block.

**Step 1: find \( \alpha \)**
Figure C.1 Spires and roughness in a rectangular test section [78]
From Equation C.2, \( \alpha \) can be found by knowing that \( C_f = 0.0032 \):

\[
\sqrt{\frac{C_f}{0.136}} = \frac{\alpha}{1 + \alpha}
\]

\[
\Rightarrow \alpha = \frac{\sqrt{\frac{C_f}{0.136}}}{1 - \sqrt{\frac{C_f}{0.136}}}
\]

\[
\Rightarrow \alpha = \frac{\sqrt{\frac{0.0032}{0.136}}}{1 - \sqrt{\frac{0.0032}{0.136}}} = \frac{0.1534}{1 - 0.1534} = 0.181
\]

Therefore, the velocity distribution at \( x = 6H \) can be determined by using Equation C.1:

\[
u = u_\delta \left( \frac{Z}{\delta} \right)\]

where \( u_\delta \) can be measured, and \( \delta \) is 20 centimeters.

**Step 2:** find \( \beta \)

From Equation C.3, \( \beta \) can be calculated by knowing \( \delta, H_o \) and \( \alpha \), i.e.,

\[
\beta = \left( \frac{\delta}{H_o} \right) \left( \frac{\alpha}{1 + \alpha} \right) = \left( \frac{20}{122} \right) \left( \frac{0.181}{1 + 0.181} \right) = 0.025
\]

**Step 3:** find \( H \)

From Equation C.4, \( H \) can be calculated by knowing \( \delta \) and \( \alpha \), i.e.,

\[
H = 1.39 \left( \frac{\delta}{1 + \frac{\alpha}{2}} \right) = 1.39 \times \left( \frac{20}{1 + \frac{0.181}{2}} \right) = 25.49 \text{ cm}
\]

**Step 4:** find \( \Psi \)

From Equation C.5, \( \Psi \) can be calculated by knowing \( \beta \) and \( \alpha \), i.e.,

\[
\Psi = \beta \left\{ \frac{2}{1 + 2\alpha} + \beta - \frac{1.13\alpha}{(1 + \alpha)(1 + \frac{\alpha}{2})} \right\} \frac{1}{(1 - \beta)^2}
\]

\[
\Rightarrow \Psi = \frac{0.025}{(1 - 0.025)^2} \left( \frac{2}{1 + (2 \times 0.181)} + 0.025 - \frac{1.13 \times 0.181}{(1 + 0.181)(1 + \frac{0.181}{2})} \right) = 0.035
\]
Step 5: find \( b \)

From Equation C.6, \( b \) can be calculated by knowing \( H, H_o \) and \( \Psi \), i.e.,

\[
\frac{b}{H} = 0.5H_o\Psi\left\{\frac{1 + \frac{2}{\psi}}{(1 + \Psi)^2}\right\} = 0.5 \times 122 \times 0.035 \times \left\{\frac{1 + \frac{0.181}{2}}{(1 + 0.035)^2 \times 20}\right\} = 0.112
\]

\[\Rightarrow b = 0.112 \times H = 0.112 \times 25.49 = 2.85 \text{ cm}\]

Step 6: find \( F \)

From Equation C.7, \( F \) can be found by knowing \( H_o, \delta \) and \( \alpha \), i.e.,

\[
F = \left\{1 + \left(\frac{\delta}{H_o}\right)\left[\frac{\alpha(3 + 2\alpha)}{1 + \alpha(1 - \frac{\delta}{H_o})}\right]\right\}^{-1} = \left\{1 + \left(\frac{20}{122}\right)\left[\frac{0.181(3 + 2 \times 0.181)}{1 + 0.181(1 - \frac{20}{122})}\right]\right\}^{-1} = 0.92
\]

Step 7: estimate \( \Delta \delta \)

If the model location is 10\( H \) downstream of \( x = 6H \), the increment of the boundary layer thickness is estimated by using Equation C.8:

\[
\Delta \delta = 0.068\alpha\left\{\frac{1 + 2\alpha}{1 + \alpha}\right\}\Delta x F
\]

\[\Rightarrow \Delta \delta = 0.068 \times 0.181 \times \left\{\frac{1 + 2 \times 0.181}{1 + 0.181}\right\} \times (10 \times 25.49) \times 0.92 = 3.33 \text{ cm}\]

so, the boundary layer thickness at \( x = 10H \) is:

\[\delta_{10H} = \delta_{6H} + \Delta \delta = 20 + 3.33 = 23.33 \text{ cm}\]

Step 8: find \( k \)

From Equation C.9, \( k \) can be calculated by knowing \( D, \delta \) and \( C_f \), i.e.,

\[
k = \delta \exp\left\{\left(\frac{2}{3}\right)\ln\left(\frac{D}{\delta}\right) - 0.1161\sqrt{\frac{2}{C_f} + 2.05}\right\}
\]

\[\Rightarrow k = 20 \times \exp\left\{\left(\frac{2}{3}\right)\ln\left(\frac{15}{20}\right) - 0.1161\sqrt{\frac{2}{0.0032} + 2.05}\right\} = 0.902 \text{ cm}\]

The solutions are: the shape of the spire is an isosceles triangle with a 25.49-centimeter height and a 12.75-centimeter base, and the spacing between two spires is 12.75 centimeters. The roughness element is a cubic block with a 0.902-centimeter length on each side. The spacing between two blocks is 15 centimeters.
This roughness-spire design technique has been employed by Thernelius [79] to simulate various flow phenomena that occur in the atmosphere as wind blows past and through typical swine production houses. A series of spires and roughness blocks have been designed and installed by Thernelius in the ISU environmental wind tunnel inlet section shown in Figure C.2. After passing the spires and blocks, the air flows past the model building which is seen at the rear of the photograph. With this setup, it is possible to simulate the atmospheric boundary layer characteristics of the flow past the actual facilities.

Figure C.2 Spires and roughness blocks installed in ISU environmental wind tunnel [79] (courtesy of Mr. Thernelius)
APPENDIX D ENVIRONMENTAL WIND TUNNEL SETUP

The configuration and the wiring diagram used in the Iowa State University environmental wind tunnel to simulate shelterbelt aerodynamics are shown in Figures D.1 and D.2. A series of spires and roughness blocks designed by using the technique introduced in Appendix C are installed from the inlet to the station ahead of the scaled shelterbelts. The shelterbelts are mounted on a disk capable of rotating in order to simulate winds from different directions. The wind tunnel inlet velocity is measured by the first pitot-static tube installed on the wall close to the inlet section. A T-stand for supporting a potentiometer controlled by a resistor is located behind the model trees. The function of the potentiometer is to control the vertical position of the second pitot-static tube. The longitudinal and lateral positions of the T-stand can be changed manually so that the windspeed in the whole flow field downstream of the shelterbelt can be measured by the second pitot-static tube. The pressure differential from the second pitot-static tube is sensed by a pressure transducer connected through an amplifier and a conditioner to a voltmeter in order to produce the digital output.

Several calibration procedures must be completed before running the wind tunnel. First, the tunnel needs to be calibrated. In other words, the relationship between the tunnel power-setting and the upstream velocity needs to be established. Second, the potentiometer needs to be calibrated to provide information between the actual vertical position of the first pitot-static tube and the reading from the resistor. Third, the output
of the digital voltmeter needs to be related to the real pressure differential measured by
the pressure transducer. Figures D.3 and D.4 display the calibration curves related to
the second and third calibration procedures. The tunnel calibration cannot be conducted
until the spires and roughness blocks are constructed and installed in the wind tunnel.

A brief introduction to the methodology of using the environmental wind tunnel
to simulate shelterbelt aerodynamics has been presented. This section is concluded by
pointing out several issues which need to be addressed. First of all, the model scaling
demands cautious and thorough study in order to satisfy similarity laws. The thickness
of the atmospheric boundary layer is approximately one kilometer, but the average
height of the artificial windbreak or the living-tree shelterbelt is about 5-15 meters.
Methods for constructing a scaled shelterbelt so that it not only satisfies the geometric
and dynamic similarities, but also is tall enough to allow sensors to differentiate the
elevation, is worthy of deliberation. Apparently, the dimension of the test section is a
major limitation for the size of the model, as is the consideration of the blockage, defined
as the ratio between the frontal area of the model and the cross-section area of the test
section. In addition, the hot-wire anemometer or Laser-Doppler Velocimetry (LDV)
technique might be a better choice than the pitot-static tube for measuring the wind
speed in the lee of the shelterbelt. Facing into the upcoming air flow is a requirement
for using the pitot-static tube to obtain accurate windspeed measurements. From the
field experiment, it is found that the wind direction behind the shelterbelt is a function
of the shelter density, incident angle of the approaching flow and the geometric position
in the wake. The uncertainty of knowing the local wind direction on the leewside of the
shelterbelt makes it more difficult to use the pitot-static tube.
Figure D.1  ISU environmental wind tunnel setup
Figure D.2  Wiring diagram of ISU environmental wind tunnel for shelter-belt analysis
Figure D.3 Calibration curve of a potentiometer

Figure D.4 Calibration curve of a voltmeter
APPENDIX E  COORDINATE TRANSFORMATION

The local spherical coordinate system with unit normal vectors $\hat{e}_\theta, \hat{e}_\phi, \hat{e}_r$ is transformed to the global coordinate $xyz$-system with unit normal vectors $\hat{i}, \hat{j}, \hat{k}$ by two successive rotations about two axes. This results in an orthogonal transformation. Figure E.1 shows the sequence of rotations. First, the spherical coordinate system rotates $-\theta$ about the $\hat{e}_\phi$ to make the $\hat{e}_r$ align with the $z$-axis. Where the $\hat{i}', \hat{j}', \hat{k}'$ represent the unit normal vectors in the coordinate system, $x'y'z'$, after the first rotation:

$$
\hat{i}' = \begin{bmatrix}
\cos \theta & 0 & \sin \theta \\
0 & 1 & 0 \\
-sin \theta & 0 & \cos \theta
\end{bmatrix} \begin{bmatrix}
\hat{e}_\theta \\
\hat{e}_\phi \\
\hat{e}_r
\end{bmatrix}
$$

(E.1)

Second, the $x'y'z'$ coordinate system rotates $-\phi$ about the $z'$-axis (or $z$-axis) to make the $x'$ and $y'$ axes match the $x$ and $y$ axes as follows:

$$
\begin{bmatrix}
\hat{i} \\
\hat{j} \\
\hat{k}
\end{bmatrix} = \begin{bmatrix}
\cos \phi & -\sin \phi & 0 \\
\sin \phi & \cos \phi & 0 \\
0 & 0 & 1
\end{bmatrix} \begin{bmatrix}
\hat{i}' \\
\hat{j}' \\
\hat{k}'
\end{bmatrix}
$$

(E.2)

Combining Equation E.1 and Equation E.2, the final transformation matrix is:

$$
\begin{bmatrix}
\hat{i} \\
\hat{j} \\
\hat{k}
\end{bmatrix} = \begin{bmatrix}
\cos \phi \cdot \cos \theta & -\sin \phi \cdot \cos \phi \cdot \sin \theta & \cos \phi \cdot \sin \theta \\
\sin \phi \cdot \cos \theta & \cos \phi \cdot \sin \phi \cdot \sin \theta & \sin \phi \cdot \sin \theta \\
-sin \theta & 0 & \cos \theta
\end{bmatrix} \begin{bmatrix}
\hat{e}_\theta \\
\hat{e}_\phi \\
\hat{e}_r
\end{bmatrix}
$$

(E.3)
By using a property of orthogonal transformations, that is, the inverse of the transformation matrix is equal to its transpose, the inverse transformation from the global $xyz$-coordinate to the local spherical coordinate is:

\[
\begin{bmatrix}
\tilde{e}_\theta \\
\tilde{e}_\phi \\
\tilde{e}_r
\end{bmatrix} =
\begin{bmatrix}
cos\phi \cos\theta & sin\phi \cos\theta & -sin\theta \\
-sin\phi & cos\phi & 0 \\
cos\phi \sin\theta & sin\phi \sin\theta & cos\theta
\end{bmatrix}
\begin{bmatrix}
\tilde{i} \\
\tilde{j} \\
\tilde{k}
\end{bmatrix}
\]  

(E.4)

Figure E.1  Relationship between two frames
APPENDIX F  LINEARIZATION OF SOURCE TERMS DUE TO DRAG FORCES

For a two-dimensional shelterbelt flow on the $x - y$ plane, the source terms from the retarding forces in the momentum equations based on typical control volumes are written as:

$$dF_x = \frac{1}{2} \rho V_{xy} \cdot (uC_d + vC_f) \cdot dA$$  \hspace{1cm} (F.1)

$$dF_y = \frac{1}{2} \rho V_{xy} \cdot (vC_d - uC_f) \cdot dA$$  \hspace{1cm} (F.2)

where the terms in the parenthesis are pressure drag and skin-friction drag coefficients, respectively.

Four cases involving different directions of the upstream flow approaching the shelterbelt are considered (see Figure F.1). Following the definitions of the aerodynamic forces acting on the shelterbelt based on the defined coordinate systems, the sign convention of the pressure drag terms in momentum equations is retained for all cases. By forcing the positive skin friction coefficient in all circumstances, the sign of the skin friction drag remains the same in case one and four, but needs to be changed for the second and third cases. The final representations of the drag terms are summarized as follows:

For $u$-momentum equation, if $u \cdot v > 0$, then

$$dF_x = \frac{1}{2} \rho V_{xy} \cdot (uC_d + vC_f) \cdot dA$$  \hspace{1cm} (F.3)

and if $u \cdot v < 0$, then

$$dF_x = \frac{1}{2} \rho V_{xy} \cdot (uC_d - vC_f) \cdot dA$$  \hspace{1cm} (F.4)
For $v$-momentum equation, if $u \cdot v > 0$, then

$$dF_y = \frac{1}{2} \rho V_{xy} \cdot (vC_d - uC_f) \cdot dA \quad (F.5)$$

and if $u \cdot v < 0$, then

$$dF_y = \frac{1}{2} \rho V_{xy} \cdot (vC_d + uC_f) \cdot dA \quad (F.6)$$

Therefore, the linearization of the source term due to the drag force can be obtained as follows:

For $u$-momentum equation, if $u \cdot v > 0$, then

$$(aP^u + \frac{1}{2} \rho V_{xy} \cdot C_d \cdot dA) u_P = \sum a_{nb} u \ u_{nb} + (b^u - \frac{1}{2} \rho V_{xy} \cdot v \cdot C_f \cdot dA) \quad (F.7)$$

and if $u \cdot v < 0$, then

$$(aP^u + \frac{1}{2} \rho V_{xy} \cdot C_d \cdot dA) u_P = \sum a_{nb} u \ u_{nb} + (b^u + \frac{1}{2} \rho V_{xy} \cdot v \cdot C_f \cdot dA) \quad (F.8)$$

For $v$-momentum equation, if $u \cdot v > 0$, then

$$(aP^v + \frac{1}{2} \rho V_{xy} \cdot C_d \cdot dA) v_P = \sum a_{nb} v \ v_{nb} + (b^v - \frac{1}{2} \rho V_{xy} \cdot u \cdot C_f \cdot dA) \quad (F.9)$$

and if $u \cdot v < 0$, then

$$(aP^v + \frac{1}{2} \rho V_{xy} \cdot C_d \cdot dA) v_P = \sum a_{nb} v \ v_{nb} + (b^v + \frac{1}{2} \rho V_{xy} \cdot u \cdot C_f \cdot dA) \quad (F.10)$$
Figure F.1 Two-dimensional shelterbelt flow cases
BIBLIOGRAPHY


