2008

Numerical Simulation of Straight Line Flow with Sparse Roughness and Tornado-like Flow on Smooth Floor

Miao Li
Iowa State University

Follow this and additional works at: https://lib.dr.iastate.edu/etd
Part of the Aerospace Engineering Commons

Recommended Citation
Graduate Theses and Dissertations. 11310.
https://lib.dr.iastate.edu/etd/11310

This Thesis is brought to you for free and open access by the Iowa State University Capstones, Theses and Dissertations at Iowa State University Digital Repository. It has been accepted for inclusion in Graduate Theses and Dissertations by an authorized administrator of Iowa State University Digital Repository. For more information, please contact digirep@iastate.edu.
Numerical simulation of straight line flow with sparse roughness and tornado-like flow on smooth floor

by

Li, Miao

A thesis submitted to the graduate faculty in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Major: Engineering Mechanics

Program of Study Committee:
Frederick L. Haan, Jr., Major Professor
Partha P. Sarkar
Tom I-Ping Shih

Iowa State University
Ames, Iowa

2008

Copyright © Li, Miao, 2008. All rights reserved.
# TABLE OF CONTENTS

**LIST OF TABLES** ................................................. iv

**LIST OF FIGURES** .............................................. iv

**ABSTRACT** .................................................. viii

**CHAPTER 1. Introduction** ...................................... 1

1.1 Straight Line Flow in Wind Tunnel ................................. 1

1.1.1 Turbulent Boundary Layer ...................................... 1

1.1.2 Effects of Roughness and Roughness Lengths .................. 2

1.2 Tornadoes .................................................. 4

**CHAPTER 2. Turbulence Model & Wall Treatments** ............... 11

2.1 Turbulence Models ........................................... 11

2.1.1 Standard $k - \varepsilon$ Model ................................. 11

2.1.2 Realizable $k - \varepsilon$ Model ................................. 12

2.1.3 Reynolds stress Model ....................................... 14

2.1.4 SST $k - \omega$ Model ....................................... 17

2.2 Near Wall Treatments ......................................... 18

2.2.1 Standard Wall Function ..................................... 19

2.2.2 Enhanced Wall Function and Two-Layer Model ................ 20

**CHAPTER 3. Numerical Simulation of Straight Line Flow on Sparse Roughnesses** 22

3.1 Overview of the cases ........................................ 22

3.1.1 Boundary Conditions ....................................... 25

3.2 The Development of Boundary Layer for Smooth Floor Cases .... 31
3.3 Boundary Layer Development for Rough Floor Cases .................................. 31
3.4 Parameter Distributions at the Outlet Boundary ........................................ 35
3.5 Distributions next to Riblets ........................................................................ 41
    3.5.1 Streamlines, Vortex Structures and T.K.E. distribution ....................... 41
    3.5.2 Flow Fields around Second Riblets relative to the First Riblet. ............... 52
3.6 Development of Turbulent Boundary Layer: Revisiting Rough Cases ............. 56

CHAPTER 4. Numerical Simulation of Tornado-like Flow ................................. 66
4.1 Overview of Various Cases: Model Construction and Boundary Conditions. ..... 66
4.2 Distributions of Key Parameters ................................................................. 70
    4.2.1 Flow Field in the Outer Region ......................................................... 70
    4.2.2 Flow Field in the Inner Region ......................................................... 71
4.3 Vortical Structures ...................................................................................... 73

CHAPTER 5. Summary and Conclusions ......................................................... 91

BIBLIOGRAPHY ......................................................................................... 93
LIST OF TABLES

Table 3.1  Boundary Conditions at the inlet. ............................................. 28
Table 3.2  Computational Domains used. ................................................. 30
Table 3.3  Conditions for Various Cases. ................................................. 30
Table 3.4  Some Parameters for Various Cases. ........................................... 42
Table 4.1  Cases V1 through V5. ............................................................. 69
Table 4.2  Boundary conditions used for tornado simulation. ....................... 69
# LIST OF FIGURES

| Figure 1.1 | Mean velocity profile. | 7 |
| Figure 1.2 | The structure of turbulent boundary layer with respect of Reynolds number by Pope [14]. | 8 |
| Figure 1.3 | Normalized Reynolds stresses $u'^2$ and $v'^2$ in terms of wall units by David B. DeGraaff and John K. Eaton [1], reproduced by Durbin [12]. | 9 |
| Figure 1.4 | Schematic of corner flow in a meridian plane by Lewellens [8]. | 10 |
| Figure 3.1 | Computational Domains. | 22 |
| Figure 3.2 | Two ways roughnesses are introduced into the computation. | 23 |
| Figure 3.3 | Grids Refinement next to the floor. | 24 |
| Figure 3.4 | $y_+$ v.s. $x$. | 26 |
| Figure 3.5 | Pressure distribution at the roof of trapezoidal and rectangular domain for Rough cases. | 27 |
| Figure 3.6 | Comparison of the boundary layer thicknesses. | 32 |
| Figure 3.7 | Mean velocities $v_x$ v.s. $y$, experimental data from Emi[5]. | 33 |
| Figure 3.8 | $C_f s$ v.s. $R_x$. | 34 |
| Figure 3.9 | “Experimental” T.K.E. are estimated as $k = \frac{1}{2} (U_{\text{mean}} I)^2$. based on data from Emi[5]. | 36 |
| Figure 3.10 | Reynolds stresses $\overline{u'w'}$, $\overline{v'w'}$ and $\overline{w'w'}$ for Cases of Rough3 and Rough4. | 37 |
| Figure 3.11 | T.K.E. v.s. $y$, the T.K.E.s for Rough3 and Rough4 are $k_{1,2} = \frac{1}{2} \left( \overline{u'w'} + \overline{v'v'} \right)$. | 38 |
| Figure 3.12 | $U_+$ v.s. $y_+$, experimental data from Emi[5]. | 39 |
| Figure 3.13 | $k_+$ v.s. $y_+$, the T.K.E.s for Rough3 and Rough4 are calculated as equation 3.3, experimental data from Emi[5]. | 40 |
Figure 3.14 Streamlines next to the first riblet. ........................................ 42
Figure 3.15 Contours of velocity magnitude $v_m$ next to the first riblet. .......... 43
Figure 3.16 T.K.E. contours next to the first riblet. .................................... 44
Figure 3.17 The velocities along the center of the SR in cases of Rough3 and Rough4. .......................... 45
Figure 3.18 Streamlines next to the first riblet: a zoom in view. .................... 46
Figure 3.19 Streamlines next to the first riblet: another zoom in view. ............. 47
Figure 3.20 Pressure distributions next to the first riblet. ............................. 48
Figure 3.21 Flood plots of vorticity $\Omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$ next to the first riblet: range from $-1.5 \times 10^3$ (white) to $-2 \times 10^4$ (black), the solid line is an isoline of the swirling strength. .......................................................... 49
Figure 3.22 Streamlines next to the first riblet. ........................................... 53
Figure 3.23 T.K.E. distributions next to the first riblet. ................................. 54
Figure 3.24 Velocity magnitudes next to the first riblet. ............................... 55
Figure 3.25 Velocity magnitudes next to the 3rd through 17th riblets in case of Rough3. ............................ 59
Figure 3.26 T.K.E.s next to the 3rd through 17th riblets in case of Rough3. ......... 60
Figure 3.27 Streamlines next to the 3rd through 17th riblets in case of Rough3. .... 61
Figure 3.28 Velocity magnitudes next to the 3rd through 17th riblets in case of Rough4. ............................ 62
Figure 3.29 T.K.E.s next to the 3rd through 17th riblets in case of Rough4. ........... 63
Figure 3.30 Velocity magnitudes next to the 3rd through 17th riblets in case of Rough4. ............................ 64
Figure 3.31 The $v_x$ profiles immediate above each riblet in cases of Rough3 ("—") and Rough4 ("–···"); the "o"s denote the experimental results; the "–···" lines denote the boundary layer thickness in case of smooth floor plus the riblet’s height, $\delta(x)_{\text{Rough}} = 0.37 \times R_x^{\frac{1}{2}} + 0.0127$ m. The dotted lines denote riblet’s height. .......................................................... 65
Figure 4.1 A schematic illustration of tornado/microburst simulator. ................ 76
Figure 4.2 The computational domain used for tornado simulation. ............... 77
Figure 4.3 The $y_+$s against the radius for cases V1 through V5. ................... 78
Figure 4.4  The azimuthal velocities $v_\theta$ at the inlet $ABCD$ and the maximum azimuthal velocities $v_{\theta\text{max}}$ ................................. 78

Figure 4.5  The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$
for case V1. .......................................................... 79

Figure 4.6  The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$
for case V2. .......................................................... 80

Figure 4.7  The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$
for case V3. .......................................................... 81

Figure 4.8  The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$
for case V4. .......................................................... 82

Figure 4.9  The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$
for case V5. .......................................................... 83

Figure 4.10  The static pressure and normalized circulation $\Omega_z$ in case V5. ................. 84

Figure 4.11  The normalized $v_\theta$ and T.K.E. in case V5. ............................................. 85

Figure 4.12  The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V1. ................. 86

Figure 4.13  The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V2. ................. 87

Figure 4.14  The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V3. ................. 88

Figure 4.15  The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V4. ................. 89

Figure 4.16  The normalized $\omega_z$ and $\Pi_2$ in case V5. ....................................... 90
ABSTRACT

The studies in this thesis simulated straight line flow and tornado-like vortex flow. The goal of the work was to develop computational simulations of flow past rough surfaces using straight-line flow as a starting point to develop simulations of tornado flows. Analysis of turbulent boundary layer development in the straight line flow was carried out both at the computational domain’s outlet, which imitate the inlet of the test section of the straight line wind tunnel and in the regions immediately next to the riblets, which were used to generate the turbulence and to accelerate the boundary layer developments. These distributions and boundary layer thickness agreed reasonably with experimental results.

The numerical distributions of velocities were studied in the tornado-like vortex flow both in the far field with large radius of maximum wind ($RMW$) and in the central region. Vortical structures in the tornado-like vortex flow, especially those in the central region were identified and studied. These structures agrees somewhat with previous numerical studies by Lewellen(1997) and Nolan(2000). New vortical structures, horizontal vortex and double vortex rolls were identified. These structures were analyzed and explained physically with respect to the development of flow field. While the project did not proceed to the point of simulating vortex flows with rough surfaces, the foundation was laid for rough-wall work to be added to these results.
CHAPTER 1. Introduction

1.1 Straight Line Flow in Wind Tunnel

The “Straight Line Flow” here refers to the atmospheric flow in the nature or the flow in the wind tunnel in the Wind Simulation and Testing (WiST) laboratory at Iowa State. The simulations in this thesis focus on the flow in front of the test section of the wind tunnel. This flow is usually used for testing the loads on buildings in atmospheric turbulent flow. So various methods are used to invoke the turbulent boundary layer so that the flow through the inlet of the test section (or equivalently, the outlet of the computational domain in this simulation) can be used to imitate the atmospheric turbulent boundary layer. One of the methods uses riblets or chains to generate the turbulence. The chains are difficult to model because of their complicated geometry. So riblets were used for the simulation. It turns out that the riblets are good substitution for the chain in simulating the experimental boundary layer thickness \( \delta \), the mean velocity profile \( u(z) \) and turbulent kinetic energy (T.K.E.) \( k \) as long as the Reynolds stress Model (RSM) were employed.

1.1.1 Turbulent Boundary Layer

Turbulent boundary layers are extensively studied in the fields of engineering and physics. Because of limited space and time, turbulence cannot be presented in all its aspects in this thesis, so what first of all will be presented here is widely accepted theory, based on which numerical results and experimental results will be compared. The mean velocity profiles for a fully turbulent flow are usually termed as “Law of Wall”, which is shown in fig. 1.1. The regime next to the wall can be basically partitioned into three layers. The layer with height from \( y_+ = 0 \) through \( y_+ \sim 5 \), where the viscosity dominates and as a result the flow is almost laminar, is called “viscous sub-layer”. In this layer, the normalized velocity in terms of “wall units” is proportional to the distance \( y \) in
2

terms of “wall units”, i.e.

\[ u_+ = y_+. \tag{1.1} \]

From \( y_+ \sim 5 \) through \( y_+ \sim 30 \), the viscosity and turbulent momentum play equally significant roles. This layer is called “buffer layer”. In the regime from \( y_+ \sim 30 \) through \( y/\delta \sim 0.3 \), turbulent momentum plays a dominant role and this layer is called “log law regime” or “log-layer”, where the normalized mean velocity has a log relation with the distances.

\[ u_+ = 2.44 \log (y_+) + 5.3. \tag{1.2} \]

The above equations 1.1 and 1.2 are in terms of “wall units” \( \ell_+ \), which is a length scale defined as

\[ \ell_+ = \frac{\nu}{u_*}, \tag{1.3} \]

\[ u_* = \sqrt{\tau_w/\rho}, \tag{1.4} \]

where \( \tau_w \) is the friction at the wall. However, the structure of the regimes next to walls can be divided in terms of both wall units and turbulent boundary layer thickness \( \delta \): the “outer layer” and “inner layer”. The outer layer begins from \( y_+ \sim 50 \); the inner layer ends up to \( y/\delta \sim 0.1 \); the layer between \( y_+ \sim 50 \) and \( y/\delta \sim 0.1 \) is in turn called the “overlap regime”. These concepts are clearly shown in fig. 1.2 from Stephen B. Pope [14].

Some of the normalized Reynolds stresses \( \bar{u'}^2 \) and \( \bar{v'}^2 \) in terms of wall units are shown in fig. 1.3. These results can be used to compare with our numerical and experimental results.

1.1.2 Effects of Roughness and Roughness Lengths

According to Schlichting [17], the major effects of roughness on mean velocity profile is that it is “pulled down” in the log and linear coordinates of \( y_+ \) and \( u_+ \). Because of wide variety of roughness, a kind of standard roughness need to be introduced to describe this effects. This kind of roughness is called “sand roughness”, which is like “spheres packed together as densely as possible on the floor”. The roughness size can be termed as \( k_s \) and is usually normalized in terms of wall units as

\[ k_s^+ = \frac{k_s}{\ell_+}. \tag{1.5} \]
So the log law can be modified as [12] and [2]

\[ u_+ = \frac{1}{\kappa} \ln \left( \frac{y}{k_s} \right) + \frac{1}{\kappa} \ln (k_s^+) + B_r (k_s^+) \]  
(1.6)

\[ = \frac{1}{\kappa} \ln \left( \frac{y}{k_s} \right) + B, \]  
(1.7)

with the additive term

\[ B \equiv \frac{1}{\kappa} \ln (k_s^+) + B_r (k_s^+). \]  
(1.8)

Then \( B \) has been measured experimentally and fit as

\[
\begin{align*}
B_r &= B; \quad k_s^+ < 2.25 \\
B_r &= \xi \left( 8.5 - \frac{1}{\kappa} \ln (k_s^+) - B \right) + B; \quad 2.25 \leq k_s^+ \leq 90 \\
B_r &= 8.5 - \frac{1}{\kappa} \ln (k_s^+); \quad k_s^+ > 90
\end{align*}
\]  
(1.9)

with the interpolation function \( \xi \) as

\[ \xi = \sin \left[ \frac{\pi}{2} \ln \left( \frac{k_s^+}{2.25} \right) - \ln \left( \frac{90}{2.25} \right) \right]. \]  
(1.10)

The equation 1.9 corresponds to the three regimes: effective smooth, transitionally rough and fully rough. The log law can be rewritten as

\[ u_+ = \frac{1}{\kappa} \ln \frac{y}{z_0}, \]  
(1.11)

where the \( z_0 = k_s \exp (-\kappa B) \) is called “hydrodynamic roughness length” in Durbin [12] and [2] or equivalently “roughness length” by Schlichting [17].

“Equivalent sand roughness” \( k_{s_{eq}} \) is defined by Schlichting [17] as

\[ k_{s_{eq}} = \exp \left\{ \kappa \lim_{y \to 0} \left[ 8.0 + \frac{1}{\kappa} \ln y - u_+(y) \right] \right\}, \]  
(1.12)

and “an experiment must be carried out to determine the velocity distribution \( u_+(y) \) in the overlap layer at the technically rough wall”. This roughness can be divided into three regimes as

\[
\begin{align*}
\text{hydraulically smooth: } & \quad 0 \leq k_s^+ \leq 5, \\
\text{transition regime: } & \quad 5 \leq k_s^+ \leq 70, \\
\text{fully rough: } & \quad 70 \leq k_s^+.
\end{align*}
\]  
(1.13)
Other than the discussion of sand roughness, riblet installed on the floor are sometimes used as roughness elements. They are often categorized as two types: “k-type” and “d-type”, which corresponds to sparsely spaced and closely spaced ribs, respectively. The sparsely spaced ribs are more discussed in chapter 3.

1.2 Tornadoes

Based on W.S. Lewellen [9], the tornado-like vortex can be divided into four regimes: core flow; surface boundary flow; central corner flow; top layer. Among these regimes, the central corner flow is most extensively studied.

The central corner flow usually takes several forms [9]: boundary layer separation, one-cell vortex, vortex breakdown or DVJ structure, two-cell vortex and multiple vortices. In this thesis, the central vortex flow takes the seemingly two-cell vortex structure as shown in chapter 4, which is also confirmed by Wei Zhang’s results [20]. The structure of the central corner flow is schematically shown in fig. 1.4 by D. C. Lewellen and W. S. Lewellen [8] for relatively higher swirl ratio. In the case of higher swirl ratio corner flow, the tornado vortex structure is often divided into four regions:

1. outer region: $\Gamma = \Gamma_\infty$ is constant;

2. surface layer: $\frac{\partial \Gamma}{\partial z}$ is large, and the vertical velocity $v_z \approx 0$;

3. upper-core region: $\frac{\partial \Gamma}{\partial r}$ is strong, radial velocity $v_r \approx 0$;

4. corner flow region: the region where the flow transitions from horizontal direction to vertical direction, and each components are significant.

This characteristic is observed too and termed as “outer region” in chapter 4. However, the flow structure of fig. 1.4 inside the $r_0$ are different than those in chapter 1.4 in that usually a inner core radius $r_i$, inside which the angular momentum is virtually zero $\Gamma \approx 0$, is defined. One of the other differences is in the structure of the surface layer in that though the contours of angular momentum
in the corner region looks similar to those in chapter 4, the velocity components and pressure don’t. The pressure distribution looks pretty cylindrical, which means that the contours are basically perpendicular to the floor and don’t have extremum at the central axis.

Lewellen [10] have also done extensive research on tornado-like vortex flow. Especially in the central corner region, he conducted a high-resolution, fully three-dimensional, unsteady simulation of the interaction of a tornado vortex with the surface. It is verified that radius of the corner flow is nearly constant with respect to increasing elevation. Instantaneous snapshots with translation also show valuable structure of vortex structures. Several updraft annulus, associated with strong secondary vortices, rotate about the primary corner vortex. Their rotating speed is much lower than that associated with maximum mean azimuthal velocities. Other than this, these secondary vortices, which are placed between the updraft and central downdraft region, are spiraling upward in a direction counter to the tornado rotation. This important and interesting phenomena is proved to be true by Wei Zhang’s streamline plots based on her experiments in case of both smooth and ground roughnesses[20].

Most of Nolan’s research concerns a given environmental forcing function that takes a form

\[
F_z = C_b \exp \left\{ - \left( \frac{r^2}{\sigma_h^2} + \frac{(z - z_{force})^2}{\sigma_v^2} \right) \right\},
\] (1.14)

where \(z_{force}\) is called the “forcing height”, \(\sigma_h\) is the “horizontal extent”, \(C_b\) is the amplitude and \(\sigma_v\) is the “vertical extent”. Other than that, Nolan used some parameters different than Lewellen’s, for example the Convective Reynolds number,

\[
Re_C = \frac{UL}{\nu}.
\] (1.15)

Vortex Reynolds number

\[
Re_V = \frac{\Gamma}{\nu} = \frac{\Omega L^2}{\nu}.
\] (1.16)

As for the solid-body simulation, \(\Omega\) is the rotation rate of the outer boundary and \(L\) is some sort of length scale. Swirl Ratio

\[
S = \frac{\Gamma r_0}{2Qh}.
\] (1.17)
Nolan noted that the $Q$ is the volume flow rate per unit axial length and $h$ is the depth of the inflow region. $r_0$ is the radius of updraft. The Internal Swirl Ratio is defined as

$$S_I = \frac{r_0}{2h_0} \int_0^{h_0} \int_0^{r_0} \frac{\Gamma(r_0, z) d\Gamma}{v_z(r, z_0) 2\pi r dr},$$

(1.18)

where $r_0$ and $h_0$ are the radius and height of the control volume that surrounds the vortex core adjacent to the surface. The Vortex Aspect Ratio is defined as

$$A_V = \frac{RMW}{ZMW},$$

(1.19)

where $RMW$ and $ZMW$ are the radius and altitude of the maximum azimuthal velocity. Basically, Nolan [11] discovered that for the low-level structure, the vortex Reynolds number $Re_V$ has controlling effect, while $Re_C$ doesn’t.

$$S_I = f(Re_V),$$

(1.20)

$$A_V = g(Re_V).$$

(1.21)

$S_I$ is a relatively arbitrary parameter, it depend on the choice of the control volume ($r_0$ and $h_0$), while $A_V$ is more practical and easier to measure with Doppler radar. Furthermore, decreasing (increasing) the rotation rate $\Omega$ and increasing (decreasing) $\nu$ have the same effect on the vortex, unless that the low viscosity allows for higher mean wind speed and a little bit more complex structure. This conclusion is kind of opposite to Le[7] and [6], where it was stated that increasing viscosity of the floor tends to increase the swirl ratio, instead of decreasing the swirl ratio by Lewellen.

The research of tornado-like vortex flow in this thesis focused on the local lower level vortical structure and comparison with corresponding experimental results presented by Haan et al. [3].

This thesis has two objectives. For the straight line flow simulation, this thesis is going to provide a reference of what’s happening between the test section and the inlet of test section of the wind tunnel. It attempted to deal with the cases of smooth floor and rough floor with chain roughnesses sparsely populated on the floor. For the tornado-like flow, the objective is to simulate the flow in the tornado/microburst simulator based on a axisymmetric and steady assumption.
Figure 1.1 Mean velocity profile.
Figure 1.2 The structure of turbulent boundary layer with respect of Reynolds number by Pope [14].
Figure 1.3 Normalized Reynolds stresses $u'^2$ and $v'^2$ in terms of wall units by David B. DeGraaff and John K. Eaton [1], reproduced by Durbin [12].
Figure 1.4  Schematic of corner flow in a meridian plane by Lewellens [8].
CHAPTER 2. Turbulence Model & Wall Treatments

2.1 Turbulence Models

This chapter is directly from Fluent User’s Guide [FLU] because the simulations are based on this part. The numerical models presented [FLU] are summarized in the following sections and subsections. The $k-\varepsilon$ models, the Reynolds Stress Model (RSM), and the Large Eddy Simulation (LES) model are the three major turbulence models or method that FLUENT uses to simulate turbulent flows. Among them, the $k-\varepsilon$ models and RSM are used in current thesis.

2.1.1 Standard $k-\varepsilon$ Model

The compressible $k-\varepsilon$ model is basically

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{2.1}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_3 \varepsilon G_b) - C_2 \varepsilon \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{2.2}
\]

where the terms

\[
G_k = -\rho u_i u_j \frac{\partial u_i}{\partial x_j}, \tag{2.3}
\]

\[
G_b = \beta g_i \frac{\mu_t}{Pr} \frac{\partial T}{\partial x_i}, \tag{2.4}
\]

\[
Y_M = 2 \rho \varepsilon M_t, \tag{2.5}
\]

\[
S_k \triangleq \text{user defined source term of } k, \tag{2.6}
\]

with the parameters

\[
\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}, \tag{2.7}
\]

\[
S = \sqrt{2S_{ij} S_{ij}}, \tag{2.8}
\]
\[ \sigma_k = 1.0, \sigma_\varepsilon = 1.3, C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09. \] (2.9)

### 2.1.2 Realizable \( k - \varepsilon \) Model

The realizable \( k - \varepsilon \) model is a “relatively recent” developed model. The term “realizable” means that the “certain mathematical” and physical constraints on the Reynolds stresses are satisfied. The realizable \( k - \varepsilon \) usually behave better for flows “involving rotation, boundary layer under strong adverse pressure gradients, separation, and recirculation”. Based on the Boussinesq Hypothesis to relate the Reynolds stresses to the mean velocity gradients:

\[
-\rho u_i'u_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \nu_t \frac{\partial u_k}{\partial x_k} \right) \delta_{ij} \tag{2.10}
\]

and the eddy viscosity definition 2.7, one of the normal Reynolds stress “in an incompressible strained mean flow” is

\[
\overline{u^2} = \frac{2}{3} k - 2 \nu_t \frac{\partial U}{\partial x},
\]

which “becomes negative, i.e., “non-realizable”, when the strain

\[
\frac{k \partial U}{\varepsilon \partial x} > \frac{1}{3C_\mu} \approx 3.7.
\]

Other than this, the Schwarz inequality for shear stresses \((u'_\alpha u'_\beta)^2 \leq u'^2_{\beta} u'^2_{\beta}\) can be violated when “the mean strain rate is too large”.

The realizable \( k - \varepsilon \) model was designed to address these “deficiencies of traditional \( k - \varepsilon \) models” by

1. “A new eddy-viscosity formula involving a variable \( C_\mu \) originally proposed by Reynolds.”

2. “A new model equation for dissipation \( \varepsilon \) based on the dynamic equation of the mean-square vorticity fluctuation.”

The modeled equations for \( k \) and \( \varepsilon \) are

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho ku_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k, \tag{2.11}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon, \tag{2.12}
\]
where

\[ C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{k}{\varepsilon}, \quad S = \sqrt{2S_{ij}S_{ij}} \]

and

\begin{align*}
G_k &= -\rho u_i u_i \frac{\partial u_j}{\partial x_i}, \\
G_b &= \beta g \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i}, \\
Y_M &= 2\rho \varepsilon M_t^2, \\
M_t &= \sqrt{k \frac{a^2}{\alpha}}.
\end{align*}

(2.13) \quad (2.14) \quad (2.15) \quad (2.16)

\( C_2 \) and \( C_{1\varepsilon} \) are constants. \( \sigma_k \) and \( \sigma_\varepsilon \) are the turbulent Prandtl Number for \( k \) and \( \varepsilon \), \( S_k \) and \( S_\varepsilon \) are user defined source terms.

The \( k \) equation 2.11 takes the same form as Standard \( k - \varepsilon \) equation as 2.1, while the \( \varepsilon \) equation doesn’t. The \( \varepsilon \) equation does not have the production of \( k \) on right-hand side, this modification results in better performance than Standard and RNG \( k - \varepsilon \) models.

Another point is that the eddy viscosity is not constant any more in the Realizable \( k - \varepsilon \) model. It is calculated as

\[ C_\mu = \frac{1}{A_0 + A_s \frac{kU^*}{\varepsilon}}, \quad (2.17) \]

where

\begin{align*}
U^* &\equiv \sqrt{S_{ij}S_{ij} + \bar{\Omega}_{ij}\bar{\Omega}_{ij}}, \\
\bar{\Omega}_{ij} &= \Omega_{ij} - 2\varepsilon_{ijk}\omega_k, \\
\Omega_{ij} &= \bar{\Omega}_{ij} - \varepsilon_{ijk}\omega_k.
\end{align*}

(2.18) \quad (2.19) \quad (2.20)

where \( \bar{\Omega}_{ij} \) is the mean rate of rotation tensor “viewed in a rotating reference frame with the angular velocity \( \omega_k \)”. The constants \( A_0 \) and \( A_s \) are

\[ A_0 = 4.04, \quad A_s = \sqrt{6} \cos \phi, \quad (2.21) \]

where

\[ \phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W), \quad W = \frac{S_{ij}S_{jk}S_{ki}}{S^3}, \quad \bar{S} = \sqrt{S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_j} \right). \]

(2.22)
2.1.3 Reynolds Stress Model

The Reynolds Stress Model (RSM) is the “most elaborate” [FLU] turbulence model that FLUENT can provide. It includes the transport equations for the Reynolds stresses and an equation for the dissipation rate. The governing equation for each of the Reynolds stresses is

\[
\frac{\partial}{\partial t} \left( \rho u'_i u'_j \right) + \frac{\partial}{\partial x_k} \left( \rho u'_i u'_j u'_k \right) = C_{ij} \equiv \text{Convection} \\
- \frac{\partial}{\partial x_k} \left[ \rho u'_i u'_k + p \left( \delta_{ij} u'_i + \delta_{ik} u'_j \right) \right] + \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial}{\partial x_k} \left( u'_i u'_j \right) \right] \\
D_{ij} \equiv \text{Turbulent Diffusion} \\
D_{L,ij} \equiv \text{Molecular Diffusion} \\
- \rho \left( u'_i u'_k \frac{\partial u'_j}{\partial x_k} + u'_j u'_k \frac{\partial u'_i}{\partial x_k} \right) - \rho \beta (g_i u'_j \theta + g_j u'_i \theta) + p \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) \\
G_{ij} \equiv \text{Buoyancy Production} \\
\phi_{ij} \equiv \text{Pressure Strain} \\
- 2\mu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} - 2\rho \Omega_k \left( u'_j u'_m \varepsilon_{ikm} + u'_i u'_m \varepsilon_{jkm} \right) \\
\varepsilon_{ij} \equiv \text{Dissipation} \\
F_{ij} \equiv \text{Production by System Rotation} \\
S_{\text{user}} \equiv \text{User-Defined Source Term}.
\]

The turbulent diffusive \( D_{ij} \) can be simplified in FLUENT as

\[
D_{ij} = \frac{\partial}{\partial x_k} \left( \mu_t \frac{\partial u'_i u'_j}{\partial x_k} \right),
\]

where the \( \mu_t \) is defined as in equation 2.7.

The pressure strain term \( \phi_{ij} \) is decomposed as

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

where the \( \phi_{ij,1} \) denotes the slow pressure strain, \( \phi_{ij,2} \) denotes the rapid pressure strain, \( \phi_{ij,w} \) denotes the wall reflection. The slow pressure strain \( \phi_{ij,1} \) is modeled as

\[
\phi_{ij,1} = -C_1 \rho \frac{\varepsilon}{k} \left( u'_i u'_j - \frac{2}{3} \delta_{ij} k \right),
\]

where \( C_1 = 1.8 \), the rapid pressure-strain term \( \phi_{ij,2} \) is modeled as

\[
\phi_{ij,2} = -C_2 \left( (P_{ij} + F_{ij} + G_{ij} - C_{ij}) - \frac{2}{3} \delta_{ij} (P + G - C) \right),
\]
where $C_2 = 0.6$, and $P_{ij}$, $F_{ij}$, $G_{ij}$, and $C_{ij}$ are defined as in equation 2.23. The $P = \frac{1}{2} P_{kk}$, $G = \frac{1}{2} G_{kk}$, and $C = \frac{1}{2} C_{kk}$. The wall-reflection $\phi_{ij,w}$ is modeled as

$$\phi_{ij,w} \equiv C_1' \varepsilon \left( \frac{u_k v_m n_k n_m \delta_{ij} - \frac{3}{2} u_i' u_k' n_j n_k - \frac{3}{2} u_j' u_k' n_i n_k}{\varepsilon d} \right) C_{kk}^{3/2}$$

$$+ C_2' \left( \phi_{km,2} n_k n_m \delta_{ij} - \frac{3}{2} \phi_{ik,2} n_j n_k - \frac{3}{2} \phi_{jk,2} n_i n_k \right) C_{kk}^{3/2} / \varepsilon d,$$

(2.28)

where $C_1' = 0.5$, $C_2' = 0.3$, $n_k$ is the $x_k$ component of the unit normal to the wall, $d$ is the least normal distance to the wall, and $C_{kk} = C_{kk}^{3/4}/\kappa$, where $C_{kk} = 0.09$ and $\kappa = 0.4187$ is the von Kármán constant. Once the RSM is involved in the enhanced wall treatment described in section 2.2, the coefficients in the pressure-strain model described in equation 2.25 through 2.28 need to be modified as functions of the Reynolds stress invariants and turbulent Reynolds number,

$$C_1 = 1 + 2.58 A \sqrt{A_2} \left\{ 1 - \exp \left[ -\left( 0.0067 \text{Re}_t \right)^2 \right] \right\},$$

$$C_2 = 0.75 \sqrt{A},$$

$$C_1' = -\frac{2}{3} C_1 + 1.67,$$

$$C_2' = \max \left( \frac{2 C_2 - \frac{1}{3}}{C_2}, 0 \right),$$

(2.29)

where the turbulent Reynolds number is defined as $\text{Re}_t = (\rho k^2 / \mu \varepsilon)$, with the $A$, $A_2$ and $A_3$ are

$$A \equiv \left( 1 - \frac{9}{8} (A_2 - A_3) \right),$$

$$A_2 \equiv a_{ik} a_{ki},$$

$$A_3 \equiv a_{ik} a_{kj} a_{ji},$$

$$a_{ij} = \left( -\frac{\rho u_i' u_j'}{\rho k} + \frac{2}{3} \rho k \delta_{ij} \right).$$

(2.30)

The Quadratic Pressure-Strain model has “superior performance” in a wide range of complex engineering flows.

$$\phi_{ij} = -(C_1 \rho \varepsilon + C_1^* P) b_{ij} + C_2 \rho \varepsilon \left( b_{ik} b_{kj} - \frac{1}{3} b_{mn} b_{mn} \delta_{ij} \right) + \left( C_3 - C_3^* \sqrt{b_{ij} b_{ij}} \right) \rho k S_{ij}$$

$$+ C_4 \rho k \left( b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij} \right) + C_5 \rho k \left( b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik} \right),$$

(2.31)

where the Reynolds stress anisotropy tensor $b_{ij}$, mean strain rate $S_{ij}$ and rate of rotation tensor $\Omega_{ij}$, are defined as

$$b_{ij} = \left( -\frac{\rho u_i' u_j'}{2 \rho k} \right),$$

(2.32)
\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right), \tag{2.33} \]

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right), \tag{2.34} \]

with constants as

\[ C_1 = 3.4, \quad C_1^* = 1.8, \quad C_2 = 4.2, \quad C_3 = 0.8, \quad C_3^* = 1.3, \quad C_4 = 1.25, \quad C_5 = 0.4. \tag{2.35} \]

For the \( k-\omega \) model, the pressure strain tensor is partitioned as

\[ \phi_{ij} = \phi_{ij,1} + \phi_{ij,2}, \tag{2.36} \]

with

\[
\phi_{ij} = -(C_1 \rho \varepsilon + C_1^* P) b_{ij} + C_2 \rho \varepsilon \left( b_{ik} b_{kj} - \frac{1}{3} b_{mn} b_{mn} \delta_{ij} \right) + \left( C_3 - C_3^* \sqrt{b_{ij} b_{ij}} \right) \rho k S_{ij} \\
+ C_4 \rho k \left( b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij} \right) + C_5 \rho k \left( b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik} \right),
\]

\[ b_{ij} = -\left( -\frac{\rho u_i'u_j'}{2\rho k} \right), \tag{2.38} \]

with constants as

\[ C_1 = 3.4, \quad C_1^* = 1.8, \quad C_2 = 4.2, \quad C_3 = 0.8, \quad C_3^* = 1.3, \quad C_4 = 1.25, \quad C_5 = 0.4. \tag{2.39} \]

Inside the domain, the turbulent kinetic energy is calculated as

\[ k = \frac{1}{2} u_i'u_i', \tag{2.40} \]

while the model equation for \( k \) is solved throughout the whole domain and used only for boundary conditions.

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \frac{1}{2} (P_{ii} + G_{ii}) - \rho \varepsilon (1 + 2M_t^2) + S_k \tag{2.41}
\]

The dissipation rate tensor \( \varepsilon_{ij} \) is calculated as

\[ \varepsilon_{ij} = \frac{2}{3} \delta_{ij} \left( \rho \varepsilon + Y_M \right), \tag{2.42} \]

with \( Y_M = 2\rho \varepsilon M_t^2 \) and turbulent Mach number \( M_t = \sqrt{\frac{k}{\alpha T}} \). The scalar dissipation rate \( \varepsilon \) is modeled

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_\varepsilon \left( \frac{1}{2} [P_{ii} + C_\varepsilon G_{ii}] \right) \frac{\varepsilon}{k} - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} + S_\varepsilon, \tag{2.43}
\]
where $\sigma = 1.0$, $C_{e1} = 1.44$, $C_{e2} = 1.92$ and $C_{3e} = \tanh \left| \frac{u}{\nu} \right|$ is the local flow direction. At the wall boundaries, the Reynolds Stress are specified in terms of $k$. Let $\tau$ denotes the tangential coordinate, $\eta$ the normal coordinate and $\lambda$ the binomal coordinates, the Reynolds stress in the cell next to the wall are calculated from

$$
\frac{u_{\tau}^2}{k} = 1.098, \quad \frac{u_{\eta}^2}{k} = 0.247, \quad \frac{u_{\lambda}^2}{k} = 0.655, \quad -\frac{u_{\tau} u_{\eta}'}{k} = 0.255.
$$

Alternatively, the Reynolds stresses can be specified in terms of wall-shear stress, like

$$
\frac{u_{\tau}^2}{u_{\tau}^2} = 5.1, \quad \frac{u_{\eta}^2}{u_{\tau}^2} = 1.0, \quad \frac{u_{\lambda}^2}{u_{\tau}^2} = 2.3, \quad -\frac{u_{\tau} u_{\eta}'}{u_{\tau}^2} = 1.0,
$$

where $u_{\tau}$ is the friction velocity $u_{\tau} \equiv \sqrt{\tau_w/\rho}$ and $\tau_w$ is the wall shear stress.

### 2.1.4 SST $k-\omega$ Model

The transport equations for the SST $k-\omega$ model are

$$
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + \tilde{G}_k - Y_k + S_k, \quad (2.46)
$$

$$
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega, \quad (2.47)
$$

where $\tilde{G}_k$ is the generation of T.K.E. by mean velocity gradients, $G_\omega$ is the generation of $\omega$. $\Gamma_k$ and $\Gamma_\omega$ are the effective diffusivity of $k$ and $\omega$, respectively. $Y_k$ and $Y_\omega$ are the dissipation of $k$ and $\omega$ by turbulence. $D_\omega$ is the cross-diffusion term. $S_k$ and $S_\omega$ are user-defined source terms. The effective diffusivities for the SST $k-\omega$ model are given by

$$
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k},
$$

$$
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega},
$$

where $\sigma_k$ and $\sigma_\omega$ are the turbulent Prandtl numbers for $k$ and $\omega$, respectively. The turbulent viscosity $\mu_t$ is

$$
\mu_t = \frac{\rho k}{\omega} \max \left[ \frac{1}{\sigma_k}, \frac{S \omega}{\sigma_k} \right],
$$

$S$ is the strain rate magnitude and

$$
\sigma_k = \frac{1}{F_1/\sigma_{k,1} + (1 - F_1)/\sigma_{k,2}},
$$

$$
\sigma_\omega = \frac{1}{F_1/\sigma_{\omega,1} + (1 - F_1)/\sigma_{\omega,2}}.
$$
\( \alpha^*, F_1 \) and \( F_2 \), are

\[
\alpha^* = \alpha^* \left( \frac{\alpha^*_0 + \text{Re}_{t}/R_k}{1 + \text{Re}_{t}/R_k} \right),
\]

(2.53)

\[
F_1 = \tanh \left( \Phi_1^4 \right),
\]

(2.54)

\[
\Phi_1 = \min \left[ \max \left( \sqrt{k}, 0.09 \omega_y \right), \frac{500 \mu}{\rho y^2 \omega} \right],
\]

(2.55)

\[
D_\omega^+ = \max \left( 2 \rho \frac{1}{\sigma_{\omega, 2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right),
\]

(2.56)

\[
F_2 = \tanh \left( \Phi_2^4 \right),
\]

(2.57)

\[
\Phi_2 = \max \left( 2 \sqrt{k}, 500 \mu \right),
\]

(2.58)

\( \tilde{G}_k \), the production of T.K.E. is defined as

\[
\tilde{G}_k = \min(G_k, 10 \rho \beta^* k \omega),
\]

(2.59)

and \( G_\omega \), the production of \( \omega \), is

\[
G_\omega = \frac{\alpha}{\nu_t} G_k.
\]

(2.60)

The dissipation rate of T.K.E. is defined as

\[
Y_k = \rho \beta^* k \omega,
\]

(2.61)

and the dissipation rate of \( \omega \) \( Y_\omega \) is

\[
Y_\omega = \rho \beta \omega^2.
\]

(2.62)

The Cross Diffusion Modification term \( D_\omega \) is calculated as

\[
D_\omega = 2 \left( 1 - F_1 \right) \rho \sigma_{\omega, 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}.
\]

(2.63)

### 2.2 Near Wall Treatments

The wall treatments for turbulent flow used in this thesis are Standard Wall Function and Enhanced Wall Function available in FLUENT [FLU]. The Standard Wall Function are “designed” to “bridge” the viscosity affected inner region and the fully turbulent outer region. It is widely used in engineering, especially high Reynolds number flows in that it is “economical, robust, and reasonably accurate”. While the Enhanced Wall Function is used to resolve the whole viscosity
dominant region based on a mesh grid paved all the way down to the wall, even with an requirement of $y_+$ (usually the “wall units” of $y_P$ are supposed to be less than one).

### 2.2.1 Standard Wall Function

The Law-of-Wall for mean velocity and Standard Wall Function are summarized as

$$ U^* = \frac{1}{\kappa} \ln(E y^*), \quad (2.64) $$

where

$$ U^* \equiv \frac{U_P C_\mu^{1/4} k_P^{1/2}}{\tau_w / \rho}, $$

$$ y^* \equiv \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu}, \quad (2.65) $$

and

$$ \kappa = \text{von Kármán constant} (= 0.4187), $$

$$ E = \text{empirical constant} (= 9.793), $$

$$ U_P = \text{mean velocity of the fluid at point P}, $$

$$ k_P = \text{turbulence kinetic energy at point P}, $$

$$ y_P = \text{distance from point P to the wall}, $$

$$ \mu = \text{dynamic viscosity of the fluid}, $$

and the log-law is employed when $y^* > 11.225$. Once the “wall units” of $y_P < 11.225$, **FLUENT** uses laminar stress-strain relationship that can be written as

$$ U^* = y^*. \quad (2.66) $$

In the $k - \varepsilon$ models and the RSM, the $k$ equation is solved in the whole interior domain, including the grids next to the walls. The boundary condition for the $k$ at the wall boundaries is

$$ \frac{\partial k}{\partial n} = 0, \quad (2.67) $$

the $G_k$ in equation 2.1.1, and the dissipation rate $\varepsilon$ at the grids point next to the walls are calculated under the condition of local equilibrium hypothesis,

$$ G_k \approx \tau_w \frac{\partial U}{\partial y} = \tau_w \frac{\tau_w}{\kappa \rho C_\mu^{1/4} k_P^{1/2} y_P}, \quad (2.68) $$
and the \( \varepsilon \) from
\[
\varepsilon_P = \frac{C_p^{3/4} \kappa^{3/2}}{\kappa y_P}. \tag{2.69}
\]

### 2.2.2 Enhanced Wall Function and Two-Layer Model

The “Enhanced Wall treatments” in FLUENT combines the “two-layer model” with an “Enhanced Wall Functions”. Based on the turbulent Reynolds number \( \text{Re}_y \)
\[
\text{Re}_y = \frac{\rho y \sqrt{k}}{\mu}, \tag{2.70}
\]
then the \( k - \varepsilon \) models or RSM are pitched with the one equation \( k - \ell \) model where
\[
\text{Re}_y = \text{Re}^*_y = 200. \tag{2.71}
\]
Above \( \text{Re}^*_y \), the \( k - \varepsilon \) models or RSM are used, while below \( \text{Re}^*_y \), one equation model is used. The turbulent viscosity \( \mu_{t,\text{enh}} \) is calculated as

\[
\mu_{t,\text{enh}} = \lambda \varepsilon \mu_t + (1 - \lambda \varepsilon) \mu_{t,\text{2layer}}, \tag{2.72}
\]
\[
\mu_{t,\text{2layer}} = \rho C \ell \mu \sqrt{k}, \tag{2.73}
\]
\[
\lambda \varepsilon = \frac{1}{2} \left[ 1 + \tanh \left( \frac{\text{Re}_y - \text{Re}^*_y}{A} \right) \right], \tag{2.74}
\]
\[
A = \frac{|\Delta \text{Re}_y|}{\tanh(0.98)}, \tag{2.75}
\]
and the \( \mu_t \) is calculated as in equation 2.7. The \( \varepsilon \) is calculated as

\[
\varepsilon = \frac{k^{3/2}}{\ell \varepsilon}, \tag{2.76}
\]
\[
\ell \varepsilon = y C \ell^* \left( 1 - e^{-\text{Re}_y/A} \right). \tag{2.77}
\]

The mean velocity is interpolated as
\[
u^+ = e^\Gamma v_{\text{lam}}^+ + e^\frac{1}{2} v_{\text{turb}}^+ \tag{2.78}
\]
with the blending function
\[
\Gamma = -\frac{a(y^+)^4}{1 + by^+}. \tag{2.79}
\]
So the derivative of equation 2.78 is
\[
\frac{dv^+}{dy^+} = e^\Gamma \frac{dv_{\text{lam}}^+}{dy^+} + e^\frac{1}{2} \frac{dv_{\text{turb}}^+}{dy^+}, \tag{2.80}
\]
with the first derivative on the right hand side calculated as

\[
\frac{du_{turb}^+}{dy^+} = \frac{1}{\kappa y^+} \left[ S'(1 - \beta u^+ - \gamma (u^+)^2) \right]^{1/2},
\]

(2.81)

where

\[
S' = \begin{cases} 
1 + \alpha y^+ & \text{for } y^+ < y_s^+ \equiv 60 \\
1 + \alpha y_s^+ & \text{for } y^+ \geq y_s^+ \equiv 60
\end{cases}
\]

(2.82)

\[
\alpha \equiv \frac{\nu_w dp}{\tau_w u^+ dx} = \frac{\mu dp}{\rho (u^+)^3 dx},
\]

(2.83)

\[
\beta \equiv \frac{\sigma_l q_w u^*}{c_p \tau_w T_w} = \frac{\sigma_l q_w}{\rho c_p u^* T_w},
\]

(2.84)

\[
\gamma \equiv \frac{\sigma_l (u^*)^2}{2c_p T_w},
\]

(2.85)

and the laminar part is modeled as

\[
u_{lam}^+ = y^+ \left( 1 + \frac{\alpha}{2} y^+ \right).
\]

(2.86)
CHAPTER 3. Numerical Simulation of Straight Line Flow on Sparse Roughnesses

3.1 Overview of the cases

This chapter summarizes the straight-line flow cases that were simulated. These cases included several different domains and several different roughness cases. This chapter first describes the geometry of the domains and the roughness and then summarizes the results of the simulations. These results are also compared to those data from Jones [5].

![Figure 3.1 Computational Domains.](image)

The cases listed here are based on several kinds of domains. The shapes of the domains are rectangular and trapezoidal. The dimensions of rectangular domains are basically $(1.5+15.8496) \times 0.6$ m or $(1.5 + 15.8496) \times 1.8$ m. The length of the trapezoidal domain is $(1.5 + 15.8496)$ m as the rect-
Figure 3.2  Two ways roughnesses are introduced into the computation.
Figure 3.3  Grids Refinement next to the floor
angular domains, with the inlet height $AB$ of 0.6 m and the outlet height $CD$ of $0.6 + \frac{1}{8} \times 0.2076$ m. The dimensions of domains are listed in table 3.2. Domain1 through Domain3 are listed in fig. 3.1.

The floor is made up of two sections: an inviscid flat-plate section from inlet $AB$ through the origin $x = 0.0$, and an viscous section extending from the origin $x = 0.0$ to the outlet $CD$. This type of configuration is proposed by Mr. Yoon and Dr. Shih [21]. In this way, “the leading-edge of the boundary-layer is resolved correctly”, which means the there will be a small vertical velocities at the leading edge from $x = 0.0$, which is closer to the practical transportation of the momentum from the boundary in the tunnel.

Rough cases and NoRough cases have different wall treatment on the second section of the floor. In the NoRough case, there are nothing at all on the floor, this is specified as zero “roughness height” when the “Standard Wall Function” is used. In the Rough cases, a roughness profile or roughness height is used to specify the roughness effect if the “Standard Wall Function” is used, and “riblets” are also used to simulate the “chain” roughness if “Enhanced Wall Function” is used. The dimension of the roughnesses is $0.5 \text{ in} \times 0.5 \text{ in}$, or $0.0127 \text{ m} \times 0.0127 \text{ m}$ equivalently, which are the dimensions of the chains (act as roughness on the floor). Seventeen roughnesses are evenly distributed from $x = 0.0$ through $x = 13.5$. The ratio of the distances between two roughnesses to the roughness size is around $\frac{13.5}{17-1} \div 0.0127 = 66.437$, which means they are very sparsely distributed. The roughnesses located at $x = 0.0 \text{ m}$ and the grids around it are shown in fig. 3.2(b). The “roughness height” or “roughness profile” is illustrated in fig. 3.2(a). This profile shows the “roughness height” is 0.0127 m between $x = 0.0 \text{ m}$ and $x = 0.0127 \text{ m}$, zero elsewhere. So the “roughness height” is actually a series of square function along the floor.

### 3.1.1 Boundary Conditions

The cases corresponds to the NoRough cases and Rough cases. The boundary conditions at the inlet $AB$ are shown in table 3.1 for various cases. The inlet velocity are given by experimental data by Emi[5], while the T.K.E. $k$ and dissipation rate $\varepsilon$ are estimated by assuming that they, along with the inlet velocity, are uniformly distributed throughout the inlet and
Figure 3.4  $y_+ \text{ v.s. } x$. 

(a) NoRough Cases.

(b) Rough Cases.
Figure 3.5 Pressure distribution at the roof of trapezoidal and rectangular domain for Rough cases.
Table 3.1  Boundary Conditions at the inlet.

<table>
<thead>
<tr>
<th></th>
<th>( U_\infty )</th>
<th>( k_\infty )</th>
<th>( \varepsilon_\infty )</th>
</tr>
</thead>
<tbody>
<tr>
<td>NoRough Cases</td>
<td>16.55</td>
<td>0.0016</td>
<td>0.1516</td>
</tr>
<tr>
<td>Rough Cases</td>
<td>16.00</td>
<td>0.0015</td>
<td>0.1415</td>
</tr>
</tbody>
</table>

1. the turbulence intensity \( I \approx 0.2\% \),

2. the turbulence viscosity \( \mu_t \approx C_\mu \rho \frac{k^2}{\varepsilon} \approx 0.1\mu \).

In this way, the T.K.E. and \( \varepsilon \) are that

\[
\begin{align*}
    k &= \frac{3}{2} (U_{\text{mean}} I)^2, \\
    \varepsilon &= 10 C_\mu \rho \frac{k^2}{\mu}.
\end{align*}
\]

The boundary condition at the first section of the floor (from \( x = -1.5 \text{ m} \) through \( x = 0.0 \text{ m} \)) are set as “inviscid wall”, by specifying the shear stress to be zeros. The boundary condition at the second section of the floor (from \( x = 0.0 \text{ m} \) to \( x = 15.8496 \text{ m} \)) are set as “viscous wall”.

Since there is no back flow and the pressure gradients are supposed to be essentially zero, the boundary condition at the CD is set as “outflow” available in FLUENT, which means that FLUENT interpolate the value at the boundary from interior. The boundary condition at the roof are set as “inviscid wall”, which is used to simulate the uniform free stream flow at higher elevation.

The conditions for the cases described in this chapter are listed in table 3.3. The \( y_1 \) in these tables is the distance of the first grid point next to the floor. The \( y_* \) is the \( y_1 \) in terms of “wall units”, i.e. \( y_* = \frac{y_1 u_*}{\nu} \), where \( u_* \) is the friction velocity. It is explained a bit more on page 41. The \( y_* \)'s are plotted in fig. 3.4(a) and 3.4(b). The oscillations in fig. 3.4(b) are caused by the roughnesses size introduced in Standard Wall Functions in Rough1 and by the riblet in Rough3 and Rough4.

The turbulence models are described in section 2.1.1; the wall treatments are in section 2.2. The “Adaption” in table 3.3 means that adaption of the cells next to the floor. One cell next to the wall may be refined or partitioned into four cells in order to have more resolution next to the wall. A
bunch of refined cells are shown in fig. 3.3.

Inevitably, if the physical domain is exactly a rectangular, the flow in this domain is not 100% zero-incidence flow, because of the boundary layer development. This means that the pressure gradients \( \frac{dp}{dx} < 0 \), the flow is actually accelerating, i.e. the free stream velocity \( U_\infty \) is increasing. The pressure drop is shown in fig 3.5(b).

In order to accomplish this, trapezoidal domains were used to accommodate the displacement thickness, which is around one eighth of the \( \delta \) calculated as in equation 3.1. Once the outlet is lifted by one eighth of the turbulent thickness, the pressure gradients \( \frac{dp}{dx} > 0 \), which is shown in fig, 3.5(a). However, even in this way, the pressure gradients aren’t exactly zero. Nevertheless, no matter we lift the outlet or not, the pressure variation all the way through the tunnel is small (\( \mathcal{O}(10) \) pascal).

Actually we cannot accomplish this unless the physical domain is exactly the same as the real wind tunnel, and we have essentially all the information at the inlet, which aren’t feasible because of limited computational resources and other reasons.
Table 3.2  Computational Domains used.

<table>
<thead>
<tr>
<th>Name</th>
<th>Shape</th>
<th>Length m</th>
<th>Inlet Height m</th>
<th>Outlet Height m</th>
<th>Riblet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain1</td>
<td>Rectangular</td>
<td>$1.5 + 15.8496$</td>
<td>0.6</td>
<td>0.6</td>
<td>Yes</td>
</tr>
<tr>
<td>Domain2</td>
<td>Trapezoidal</td>
<td>$1.5 + 15.8496$</td>
<td>0.6</td>
<td>$0.6 + \frac{1}{5} \times 0.2076$</td>
<td>No</td>
</tr>
<tr>
<td>Domain3</td>
<td>Rectangular</td>
<td>$1.5 + 15.8496$</td>
<td>1.8</td>
<td>1.8</td>
<td>Yes</td>
</tr>
<tr>
<td>Domain4</td>
<td>Rectangular</td>
<td>$1.5 + 15.8496$</td>
<td>1.8</td>
<td>1.8</td>
<td>No</td>
</tr>
</tbody>
</table>

Table 3.3  Conditions for Various Cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>Domain</th>
<th>$y_1$</th>
<th>$y_+^*$</th>
<th>Cell #</th>
<th>Turbulence Model</th>
<th>Wall Treatment</th>
<th>Roughness</th>
<th>Adaption</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rough1</td>
<td>Domain2</td>
<td>$10^{-3}$</td>
<td>~ 30</td>
<td>315,000</td>
<td>Standard $k - \varepsilon$</td>
<td>Standard Wall Function</td>
<td>Roughness Size</td>
<td>No</td>
</tr>
<tr>
<td>Rough2</td>
<td>Domain2</td>
<td>$2 \times 10^{-4}$</td>
<td>~ 5</td>
<td>450,625</td>
<td>Standard $k - \varepsilon$</td>
<td>Standard Wall Function</td>
<td>Roughness Size</td>
<td>No</td>
</tr>
<tr>
<td>Rough3</td>
<td>Domain1</td>
<td>-</td>
<td>~ 1.5</td>
<td>1642,775</td>
<td>Reynolds stress Model</td>
<td>Enhanced Wall Function</td>
<td>Riblet</td>
<td>Yes</td>
</tr>
<tr>
<td>Rough4</td>
<td>Domain3</td>
<td>-</td>
<td>~ 1.5</td>
<td>1819,525</td>
<td>Reynolds stress Model</td>
<td>Enhanced Wall Function</td>
<td>Riblet</td>
<td>Yes</td>
</tr>
<tr>
<td>NoRough1</td>
<td>Domain2</td>
<td>$10^{-3}$</td>
<td>~ 20</td>
<td>315,288</td>
<td>Realizable $k - \varepsilon$</td>
<td>Standard Wall Function</td>
<td>-</td>
<td>No</td>
</tr>
<tr>
<td>NoRough2</td>
<td>Domain2</td>
<td>-</td>
<td>~ 1</td>
<td>688,788</td>
<td>Realizable $k - \varepsilon$</td>
<td>Enhanced Wall Function</td>
<td>-</td>
<td>Yes</td>
</tr>
<tr>
<td>NoRough3</td>
<td>Domain4</td>
<td>-</td>
<td>~ 1</td>
<td>1121,332</td>
<td>Standard $k - \varepsilon$</td>
<td>Enhanced Wall Function</td>
<td>-</td>
<td>Yes</td>
</tr>
</tbody>
</table>
3.2 The Development of Boundary Layer for Smooth Floor Cases

The results of this development for smooth case agree pretty well with those by boundary layer theory. The turbulent boundary layer thickness is basically calculated through \[ \delta(x)|_{x=15.8496} = 0.37 x \left( \frac{U_\infty}{\nu} \right)^{-\frac{1}{2}} \bigg|_{x=15.8496} = 0.37 x R_x^{-\frac{1}{2}} \bigg|_{x=15.8496} = 0.2076 \text{ m}, \]
which is consistent with our experimental and numerical results as shown in fig 3.6(a), 3.6(b) and 3.7(a). As we can see, the thickness \( \delta \) at the outlet is around 0.2 m.

Another parameter that agrees somewhat with the theoretical one is the skin friction coefficient \( C_f \), which is shown in fig 3.8, with \( R_x = \frac{U_\infty x}{\nu} \). The various coefficients are defined as

\[
C_f = 0.0576 R_x^{-\frac{1}{7}}, \\
C_f = 0.370 [\ln (R_x)]^{-2.584}, \\
C_f = 0.02294 R_x^{-0.139}, \\
C_f = [2 \ln (R_x) - 0.65]^{-2.3}.
\]

Among them, the numerical results agree fairly well with the \( C_f = 0.0576 R_x^{-\frac{1}{7}} \).

3.3 Boundary Layer Development for Rough Floor Cases

In the Rough cases, the free stream velocity \( U_\infty = 16.00 \text{ m/s} \), as shown in table 3.1, which is little bit less than that of smooth case. So the boundary layer at the outlet is definitely thicker than that of NoRough cases. If there is no roughnesses on the floor, the \( \delta \) for \( U_\infty = 16.00 \text{ m/s} \) is obtained as

\[
\delta(x)|_{x=15.8496} = 0.37 x \left( \frac{U_\infty}{\nu} \right)^{-\frac{1}{2}} \bigg|_{x=15.8496} = 0.37 x R_x^{-\frac{1}{2}} \bigg|_{x=15.8496} = 0.2091 \text{ m}.
\]
This is much smaller than the numerical and experimental thickness for the Rough cases, which is almost 0.4 m. This indicate that the chain roughnesses, although they are very sparsely distributed on the floor, still have tremendous effects on the boundary layer development. The thickness \( \delta \) can be detected in fig 3.7(b). As we can see, the experimental data by Emi[5] and the simulation with “riblets” on the floor correctly estimated the thickness, while the simulation with “standard wall function” underestimated the thickness.
Figure 3.6 Comparison of the boundary layer thicknesses.
Figure 3.7  Mean velocities $v_x$ v.s. $y$, experimental data from Emi[5].
Figure 3.8 \( C_f \) v.s. \( R_x \)
3.4 Parameter Distributions at the Outlet Boundary

The distributions of mean velocities at the outlet are shown in fig. 3.7(a) and 3.7(b). The numerical simulations agree to a great extent with the experimental ones.

Since lack of T.K.E. information at the outlet (the data available are \( U_{\text{mean}}, u_{\text{RMS}}, \text{skewness, turbulence intensity } I, \text{ etc.} \)), it is hard to compare itself directly with existing numerical data. Even though, it’s still possible to make an estimation. Suppose the fluctuation velocity \( v' \) consists of three components, \((u', v', w')\). Then the Reynolds stress \( u'u' \), after developed along the wind tunnel, is much greater than the Reynolds stresses \( v'v' \) and \( w'w' \). This assumption is valid somewhat near the wall [13]. Then the T.K.E. is estimated as

\[
k = \frac{1}{2} \left( U_{\text{mean}} I \right)^2.
\]

(3.2)

The estimated experimental T.K.E. is compared with numerical ones in fig. 3.9(a). We can see that the estimated T.K.E.s are matched pretty well with the numerical ones. However, the “Experimental” T.K.E. from equation 3.2 does not agree with those of Rough3 and Rough4. So we need more consideration about these discrepancies.

Three Reynolds stresses, \( uu', vv' \) and \( ww' \), are shown in fig. 3.10(a) and 3.10(b). As we know, the hot-wire probes may not be sensitive to all of the Reynolds stresses. Once the two pins of a hot-wire sensor are aligned with the span-wise \((w')\) direction, they must have poor response to the span-wise fluctuation velocity \( w' \). Thus we suppose that the probes are sensitive only to the Reynolds stresses \( uu' \) and \( vv' \) but not \( ww' \). Then the numerical T.K.E. should be computed in the similar way

\[
k_{1,2} = \frac{1}{2} \left( uu' + vv' \right),
\]

(3.3)

instead of T.K.E.’s original definition \( \frac{1}{2} \left( uu' + vv' + ww' \right). \) The “Experimental” T.K.E. and numerical \( k_{1,2} \) are illustrated in fig. 3.11. They are apparently matched more with each other.

The normalized version of the figures in this chapter are listed below. The normalized mean velocity
Figure 3.9 “Experimental” T.K.E. are estimated as $k = \frac{1}{2} (U_{\text{mean}} l)^2$. Based on data from Emi[5].
Figure 3.10  Reynolds stresses $u' u'$, $v' v'$ and $w' w'$ for Cases of Rough3 and Rough4.
Figure 3.11  T.K.E. v.s. $y$, the T.K.E.s for Rough3 and Rough4 are $k_{1,2} = \frac{1}{2} \left( \overline{u'u'} + \overline{v'v'} \right)$. 
Figure 3.12 $U_+ \text{ v.s. } y_+$, experimental data from Emi[5].
Figure 3.13  \( k_+ \) v.s. \( y_+ \), the T.K.E.s for Rough3 and Rough4 are calculated as equation 3.3, experimental data from Emi[5].
and T.K.E. in terms of “wall units” are listed in fig. 3.12 and 3.13. As can be seen in figures 3.7(b) and 3.12(b), it is apparent that the “Enhanced Wall Function”, “riblets” and “Wall Adaption” are essential to the simulation of the roughness effects. The boundary layer thickness, velocity profiles and T.K.E. predicted by the Rough3 and Rough4 matched much better with experimental results and DNS results than those by Rough1 and Rough2. Another point to note is that the “riblets” turn out to be a good approximation to the “chain” roughnesses used in the wind tunnel.

Some parameters concerning these cases are listed in table 3.4. They are defined correspondingly

\[ u_* = \sqrt{\frac{\tau_{wall}}{\rho}}, \]  
\[ \ell_+ = \frac{\nu}{u_*}, \]  
\[ y_+ = \frac{y}{\ell_+}, \]  
\[ U_+ = \frac{U}{u_*}, \]  
\[ k_+ = \frac{k}{u_*^2}, \]  
\[ \delta = \int_0^\infty \left(1 - \frac{u}{U_\infty}\right) dy, \]  
\[ \theta = \int_0^\infty \left(1 - \frac{u}{U_\infty}\right) \left(\frac{u}{U_\infty}\right) dy, \]  
\[ R_\delta = \frac{U_{bulk} \delta}{\nu}, \]  
\[ R_\theta = \frac{U_{bulk} \theta}{\nu}, \]

where the \( u_* \) is the friction velocity, the \( \tau_{wall} \) is the friction force on the wall. Technically the friction velocity is interpolated from the velocity profile. The \( \ell_+ \) is the “wall unit” used to normalize the distances next to the wall. The \( \delta \) and \( \theta \) are the displacement thickness and momentum thickness, respectively.

3.5 Distributions next to Riblets

3.5.1 Streamlines, Vortex Structures and T.K.E. distribution

Fig. 3.15 shows the contours of velocity magnitude \( v_m = \sqrt{v_x^2 + v_y^2} \). As we can see, stemming from the upstream upper corner is a shear layer that separate the upper free stream and the separation-
Table 3.4  Some Parameters for Various Cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>$u_\ast$ (m/s)</th>
<th>$\ell_\ast$ ($10^{-5}$ m)</th>
<th>$U_{bulk}$ (m/s)</th>
<th>$\theta$ m</th>
<th>$\delta$ m</th>
<th>$Re$</th>
<th>$Re_\delta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rough1</td>
<td>0.5492</td>
<td>2.6599</td>
<td>15.1582</td>
<td>0.0222</td>
<td>0.0285</td>
<td>23044</td>
<td>29616</td>
</tr>
<tr>
<td>Rough2</td>
<td>0.6177</td>
<td>2.3648</td>
<td>15.3795</td>
<td>0.0283</td>
<td>0.0385</td>
<td>29800</td>
<td>39518</td>
</tr>
<tr>
<td>Rough3</td>
<td>0.5322</td>
<td>2.7445</td>
<td>17.9167</td>
<td>0.0473</td>
<td>0.0642</td>
<td>57967</td>
<td>78736</td>
</tr>
<tr>
<td>Rough4</td>
<td>0.4847</td>
<td>3.0137</td>
<td>16.6443</td>
<td>0.0517</td>
<td>0.0697</td>
<td>58945</td>
<td>79400</td>
</tr>
<tr>
<td>Chain</td>
<td>0.4680</td>
<td>3.1207</td>
<td>16.0066</td>
<td>0.0460</td>
<td>0.0631</td>
<td>50384</td>
<td>69176</td>
</tr>
<tr>
<td>NoRough1</td>
<td>0.6108</td>
<td>2.3913</td>
<td>16.6355</td>
<td>0.0201</td>
<td>0.0256</td>
<td>22756</td>
<td>29009</td>
</tr>
<tr>
<td>NoRough2</td>
<td>0.5936</td>
<td>2.4608</td>
<td>16.5477</td>
<td>0.0204</td>
<td>0.0260</td>
<td>23076</td>
<td>29449</td>
</tr>
<tr>
<td>NoRough3</td>
<td>0.5872</td>
<td>2.4874</td>
<td>16.2352</td>
<td>0.0208</td>
<td>0.0260</td>
<td>23066</td>
<td>29009</td>
</tr>
<tr>
<td>Clean</td>
<td>0.6008</td>
<td>2.4312</td>
<td>16.6376</td>
<td>0.0263</td>
<td>0.0336</td>
<td>30006</td>
<td>38227</td>
</tr>
</tbody>
</table>

Figure 3.14  Streamlines next to the first riblet.
Figure 3.15  Contours of velocity magnitude $v_m$ next to the first riblet.
Figure 3.16  T.K.E. contours next to the first riblet.
Figure 3.17 The velocities along the center of the SR in cases of Rough3 and Rough4.
Figure 3.18  Streamlines next to the first riblet: a zoom in view.
Figure 3.19 Streamlines next to the first riblet: another zoom in view.
(a) Rough3

(b) Rough4

Figure 3.20  Pressure distributions next to the first riblet.
Figure 3.21 Flood plots of vorticity $\Omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$ next to the first riblet: range from $-1.5 \times 10^3$ (white) to $-2 \times 10^4$ (black), the solid line is an isoline of the swirling strength.
reattachment (SR) region downstream of the riblet. The upper free stream velocity is about 20 m/s, while the velocity beneath the shear layer, is just about 1 m/s. The velocity magnitude changes a lot across the shear layer, with the thickness of just around 0.02 m as shown in fig. 3.17. This shear layer is one of the source of the T.K.E., which is turned out to be true in fig. 3.16 that the maximum value of T.K.E. occurs in the shear layer region. Other than this, this shear layer has seemingly vortex shedding structure. “Seemingly” means that this periodic structure shown in Rough3 is not shown as in case of Rough4. However, this structure is reasonable because the shear layer structure is not stable enough to maintain itself due to the large value of the Reynolds number in case of Rough3 and Rough4. The Reynolds number is basically \( Re = \frac{U h}{\nu} = \frac{16 \times 0.0127}{1.46 \times 10^{-5}} = 1.391 \times 10^4 \), which will definitely cause the shear layer to be unstable.

Fig. 3.14 shows the streamlines next to the first riblet, i.e. the flow pattern. Though Rough3 and Rough4 cases have different geometries, they illustrate some similar structures to each other. There is no SR regions developed in the upstream of the riblet. These results are somewhat consistent to Ikeda and Durbin [4], which does not shows SR in front of the riblet predicted by steady state RANS \((\nu f - f)\). Other than this, there is a primary SR region developed in the downstream region of the riblet. However, primary SR regions for Rough3 and Rough4 cases have different sizes and locations. The Rough3 primary SR region has its reattachment around \( x = 0.23 \) m on the floor, while the Rough4 at \( x = 0.17 \) m. This means that the primary SR for Rough3 is relatively elongated. This may be due to the greater bulk velocity in Rough3, or equivalently more effects from a much lower roof.

Though the primary SRs in the downstream region have different sizes, they have seemingly similar structure. Both of the primary SRs have a very strong shear layer stemming from the upper left corner at \((x, y) = (0, 0.0127)\) m. Another similarity between these two flow patterns is that both of them have a much smaller secondary SR in the riblet’s immediate downstream region. This is also consistent to Ikeda and Durbin. Furthermore, these two smaller secondary SR have essentially same dimensions. Their reattachments occur at \( x = 0.042 \) m on the floor and \( y = 0.008 \) m on the riblet’s downstream side.
The difference between the sizes of primary SR may be due to the difference between two bulk velocities. Suppose that the primary SR center in Rough3 located at $x = 0.10 \text{ m}$ and at $x = 0.07 \text{ m}$ in Rough4, respectively. Fig. 3.17 shows the $v_x$ through the centers of the primary SRs for the two cases. If the core size of the SR is defined at the height of the maximum $v_x$ occurs, then the maximum $v_x (v_{x\max})$ of Rough3 is greater than that of Rough4, i.e. $v_{x\max} \text{Rough3} > v_{x\max} \text{Rough4}$. Furthermore, the bulk velocity $v_{x\text{bulk}}$ of Rough3, which is the asymptotic value when $y \to \infty$, is always greater than that of Rough4, which is due to the higher roof of Rough4. The height of Rough3 above the first riblet is 0.6 m, which is much less than the height of Rough4, 1.8 m. So once the flow encounters an abrupt change of channel’s width (the first riblet), the bulk velocity of Rough3 must be more sensitive to that of Rough4, and in turn, becomes greater than that of Rough4 as shown in fig. 3.17. This difference is retained all the way from higher elevation to primary SR’s upper boundary as shown in fig. 3.17, from where the SR is elongated.

Other than the two SRs in the downstream regions of these two riblet, there is seemingly a much smaller SR on top of the riblet, as shown in fig. 3.18. This small SR is reported by Leonardi [18] in case of the ratio of distance to roughness size $\frac{\frac{w}{h}}{\frac{h}{n}} \geq 7$, though their SRs look much thinner.

Fig. 3.21 shows the flood plots of vorticity $\Omega = \frac{\partial u}{\partial x} - \frac{\partial u}{\partial y}$ and an isoline of swirling strength defined as $\lambda_2$,

$$
\lambda_2 = \left( \text{trace } D^{2-D} \right)^2 - 4 \det D^{2-D} \\
= \left( \frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} \right)^2 - 4 \left( \frac{\partial U}{\partial x} \frac{\partial V}{\partial y} - \frac{\partial U}{\partial y} \frac{\partial V}{\partial x} \right),
$$

$$
D^{2-D} = \begin{pmatrix}
\frac{\partial U}{\partial x} & \frac{\partial V}{\partial x} \\
\frac{\partial U}{\partial y} & \frac{\partial V}{\partial y}
\end{pmatrix}.
$$

Though fig. 3.21(a) and fig. 3.21(b) look different in their shapes, they still have a couple of things in common. First of all, the maximum vorticity occurs at the upstream side and the tiny portion of the top of the riblet before the separation. This is physically reasonable since there is non-slip condition used on the floor in front of first riblet, and the stress there must be very big. Another
important vortex structure is the tiny vortex on the upper right corner of the riblet. This vortex is actually hinted in fig. 3.18 as well by the tiny enclosed streamline contours above there.

Fig. 3.21(a) and fig. 3.21(b) show different vortex structures above the riblet. Fig. 3.21(a) demonstrates obvious vortex shedding structure stemming from the upstream corner of the riblet. This vortex shedding is hinted in the streamline in fig. 3.19(a) and pressure distribution in fig. 3.20(a). The streamlines are closed periodically above the riblet, the pressure contours show also periodical pressure oscillation and lower pressure regions. The three lower pressure regions coincides not only with the enclosed streamline, but also the isolines of swirling strength $\lambda_2$. The swirling strength $\lambda_2$, proposed by Adrian [15], is a powerful tool once combined with the vorticity contours, streamlines and other contours like pressure to identify the vortices. Other than this, the swirling strength can discriminate against vorticity caused by the strong shear at boundary. This is illustrated in fig. 3.21(a), we can see that the centers of the lower pressure region, the centers of the isoline of swirling strength and the centers of the lower vorticity regions overlap with each other very well, they almost coincide. Unfortunately, this vortex shedding is not confirmed in fig. 3.21(b), the isoline of swirling strength is tremendously elongated along the shear layer and enclosed streamlines are not observed, though several lower pressure region are obvious in fig. 3.20(b).

Fig. 3.16 shows the T.K.E. distributions. Both of them shows that the T.K.E. are produced across the shear layer, this is physically reasonable even the T.K.E. budget is not available in FLUENT. However, even in the downstream primary SR region, the flow is highly turbulent if the T.K.E. distribution is trustable. If look at the region around $(x, y) = (0.1, 0.01)$, the magnitude of RMS velocity $v_{RMS} \sim \mathcal{O}(\sqrt{k}) \sim \mathcal{O}(1)$, while the mean velocity there is just around $0.74 \text{ m/s}$, which means that the turbulence intensity exceeds 100%. This indicate that the flow in the downstream SR region is highly turbulent, or in another word, highly unsteady.

3.5.2 Flow Fields around Second Riblets relative to the First Riblet.

An important difference between the flow fields of first and second riblet is the vortex structure in the upstream, downstream region and on top of other riblets.
Figure 3.22 Streaklines next to the first riblet.
Figure 3.23  T.K.E. distributions next to the first riblet.
Figure 3.24 Velocity magnitudes next to the first riblet.
As can be seen in fig. 3.22, there is a SR in the upstream region of the second riblet, comparing to that in fig. 3.18(a). This may be due to the viscous effects on the floor between the first and second riblets. Since this effects, the mean velocity profile of the oncoming flow in front of the second riblet is not that uniform as that in front of the first riblet. The nature of this difference of the two oncoming flow profile is actually the difference between the momentum of the two flow. This means because of the friction effects accumulated through first to the second riblet, the momentum of the flow next to the wall in front of the second riblet has been dissipated. Or equivalently, this dissipated momentum is supplemented by the momentum imported from the inlet AB. So intuitively, in front of the second riblet, the flow does not have enough momentum to climb over the roughness, instead of wandering around in the upstream corner, and result in a small SR there.

On top of the second riblet, there is no obvious vortex shedding and other periodic structures (such as lower pressure regions, enclosed streamlines, or even isoline of swirl strength). The primary SR, no matter in Rough3 or Rough4, has the same sizes; horizontal span is of 0.85 m through of 0.95 m and the vertical span is from the floor through up to 0.02 m. Not only the streamline in fig. 3.22, but also the T.K.E. in fig. 3.23 and velocity magnitude $v_m$ shows that the flow fields of second riblet for Rough3 and Rough4 are matched much better with each other than those of first riblet. So in general, the flow structure around the second riblet of Rough3 and Rough4 are essentially same as each other.

### 3.6 Development of Turbulent Boundary Layer: Revisiting Rough Cases

Because of the limited time and space, the flow fields around each riblet can't be analyzed. Fig. 3.25 and 3.28 is the contour plots of velocity magnitude around the third through eighth riblets. As can be seen, the development of the boundary layer is apparent through these figures. The higher speed region is moving upward. Or in another word, the riblets are more and more overwhelmed in the low speed region.
However, fig. 3.26 and 3.29 show the T.K.E. contour plots. These plots show that the T.K.E. distributions have identical structure, while the magnitude is bit different. With the development of the boundary layer, the magnitude of T.K.E. is a bit decreasing, which means that more a riblet is located closer to the leading edge, more it contributes to the boundary layer development.

Fig. 3.27 and 3.30 shows the streamlines. These streamlines have similar structure with those in fig. 3.22(a). Both have upstream small SRs, downstream secondary SRs and downstream primary SRs. These SRs, especially the primary SRs have basically the same dimension, i.e. horizontal span of around 0.1 m and vertical span of around 0.2 m. On top of these riblets, the streamlines do not show any small SR or bubbles as the streamlines do on top of the first riblet in fig. 3.18(a). Their structure are more similar or consistent to the structure on top of the second riblet in fig. 3.19(b).

Based on the streamlines in fig. 3.14 and fig. 3.22, it is very likely that the flow field around first riblet is periodic. Vortex shedding stemming from the corner occurs above the first riblet. These vortices, as in fig. 3.21(a), is further transported upward and downstream. Finally, they are dissipated in the upper free stream. What’s happening above the second riblets is not clear yet, the sub-figures in fig. 3.22 have different structure above the riblet, so it needs more investigation.

Fig. 3.31 is the \(v_x\) profiles above each riblet (denoted by \(i = 1, \ldots, 17\)) in case of Rough3 (“—”) and Rough4 (“–·–”); the “–·–” line is the boundary layer thickness in case of smooth floor to compare with Rough cases; “◦”s denotes the experimental results. As can be seen, though not very clearly, the differences between the two thickness \(\delta_{\text{Rough}} - \delta_{\text{NoRough}}\) varies along the floor. In case of the Rough cases, the boundary layer thickness \(\delta(x)_{\text{Rough}}\) increases much faster than \(\delta(x)_{\text{Rough}}\) does around the leading edge. This may be due to the much stronger effects caused by the first a couple of riblets. In the middle and downstream region of the domain, the difference does not vary much, it stays pretty stable. So basically, the roughness effects caused by the riblets are more contributed by the riblets in the upstream region of the whole domain instead of those in the downstream region. This is confirmed in fig. 3.25 and 3.28 in that the riblets downstream are overwhelmed in
the low speed region. Since the low momentum in the flow, the riblets don’t affect the flow that much by dissipating flow momentum as the upstream riblets do.
Figure 3.25 Velocity magnitudes next to the 3rd through 17th riblets in case of Rough3.
Figure 3.26 T.K.E.s next to the 3d through 17th riblets in case of Rough3.
Figure 3.27 Streamlines next to the 3d through 17th riblets in case of Rough3.
Figure 3.28  Velocity magnitudes next to the 3rd through 17th riblets in case of Rough4.
Figure 3.29  T.K.E.s next to the 3rd through 17th riblets in case of Rough4.
Figure 3.30  Velocity magnitudes next to the 3rd through 17th riblets in case of Rough4.
Figure 3.31 The $v_x$ profiles immediately above each riblet in cases of Rough3 ("—") and Rough4 ("–·"); the “◦”s denote the experimental results; the “–··” lines denote the boundary layer thickness in case of smooth floor plus the riblet’s height, $\delta(x)_{\text{Rough}} = 0.37 x R_x^{-\frac{4}{5}} + 0.0127 \text{ m}$. The dotted lines denote riblet’s height.
CHAPTER 4. Numerical Simulation of Tornado-like Flow

4.1 Overview of Various Cases: Model Construction and Boundary Conditions.

This chapter summarizes the Tornado-like flow cases that were simulated. The cases simulated were based on the V1 through V5 cases experimentally considered by Haan et al. [3]. This chapter first describes the geometry of the domains and then summarizes the results of the simulations.

The tornado simulator is shown schematically in fig. 4.1. Based on their configuration, the computational domain was constructed as shown in fig. 4.2. The whole domain was constructed basically as a pie grid, i.e. $\frac{1}{8}$ of the whole experimental cylinder domain with the angle of $\frac{\pi}{4}$. Periodic boundary conditions were used to simulate the tornado-like flow based on the axisymmetric assumption. In fig. 4.2(a), the surface $ABCD$ is a “velocity-inlet” boundary, which is used to specify circulations and mass flow rates. The channel $ABCDEFGH$ is used to direct the flow into the domain to generate the vortex revolving around the axis. In fig. 4.2(b), the surface $IJK$ is the outlet, where the flow goes upward out of the domain. The surfaces $BCKJ$ and $ADKI$ are the periodic boundaries. The experimental cases are listed in table 4.1. Listed in table 4.1 are also some numerical conditions and results, such as $y_+$, the radius of maximum wind ($RMW$), height of maximum wind ($HMW$) $v_{\theta_{\text{max}}}$. The case names in this thesis corresponds to those used in Haan et al. [3]. The $Q$ is the mass flow rate passing through the surface $ABCD$. $v_r$ and $v_\theta$ are the evenly distributed components of the velocity passing through the surface $ABCD$. $y_+$ is the least distance in “wall units” at the floor. $v_{\theta_{\text{max}}}$ is the maximum azimuthal velocity at lower elevation near the floor. $RMW$ and $HMW$ are the radius and height where the $v_{\theta_{\text{max}}}$ occurs. The number of cells for each case is
2,943,756. Because of limited computational resources, the grids have been constructed such that the $y_+$ is no more than 200, so that the wall function is kind of valid for all of calculations the five cases. The $y_+$ were plotted against radius to show that their values are no more than 200 in Fig. 4.3.

The boundary conditions for the surfaces were simple, and are listed in table 4.2. The major solver for this simulation is basically an incompressible solver, so at the “velocity inlet” boundary $ABCD$, velocity components must be specified. Cylindrical components are set to match the circulation and mass flow rate listed by Haan et al. [3]. The components were set evenly through the whole surface and defined as

$$v_r = \frac{Q}{S},$$
$$v_\theta = v_r \tan \theta,$$

where the $\theta$ is the “Vane Angle”, $S$ is the 8th times the area of surface $ABCD$. In addition, turbulence parameters need to be set at the inlet boundary. “Turbulence Intensity” and “Turbulent viscosity Ratio” were chosen to specify the turbulence as

$$I = 5\%,$$
$$\frac{\mu_t}{\mu} = 0.1.$$
faces and the azimuthal velocities $v_z$ and radial velocities $v_r$ were zero on the axis $KO$.

The surface $MNGH$ were set as “inviscid wall” because

1. the “suction” produced by the honeycomb is hard to simulate to force the flow out of the domain;

2. the open domain does not have any viscous effect on the flow;

3. the overall mass flow rate should be zero across the open boundary.

Since the “suction” effect is hard to imitate, there should be other ways to force the flow coming in between the two control cylinders and going out of the domain through $IJK$, so a wall was set at the boundary to enforce the zero mass flow rate also. It was set “inviscid” since there is no viscous effect at the boundary. The floor $OHG$ was to imitate the platform of the tornado/microburst simulator, so it was set as “viscous wall” in FLUENT. Surface $IJK$ was set as “outflow” condition and was the “only” outlet of the whole domain. “Outflow” means that the values of all parameters were interpolated from the interior domain, which means this boundary does not act as a suction. This is why the open boundary at the outer edge of the floor was closed by an inviscid wall.

The turbulent models used were basically $k - \varepsilon$ model and shear-stress transport (SST) $k - \omega$ model, which were described in more details in section 2.1.2 and section 2.1.4. The wall treatments used in various wall boundary were “Standard Wall Function”, with the “roughness height” specified as zero.
Table 4.1 Cases V1 through V5.

<table>
<thead>
<tr>
<th>Case</th>
<th>Vane Angle</th>
<th>Q (m$^3$/s)</th>
<th>$v_r$ (m/s)</th>
<th>$v_\theta$ (m/s)</th>
<th>$y_+$</th>
<th>$v_{r \text{max}}$ (m/s)</th>
<th>$v_{\theta \text{max}}$ (m/s)</th>
<th>RMW m</th>
<th>HMW m</th>
</tr>
</thead>
<tbody>
<tr>
<td>V1</td>
<td>15</td>
<td>14.4</td>
<td>3.5242</td>
<td>0.9443</td>
<td></td>
<td>500.3869</td>
<td>4.8672</td>
<td>10.1777</td>
<td>0.12991</td>
</tr>
<tr>
<td>V1</td>
<td>15</td>
<td>14.4</td>
<td>3.5242</td>
<td>0.9443</td>
<td></td>
<td>500.3869</td>
<td>4.8672</td>
<td>10.1777</td>
<td>0.12991</td>
</tr>
<tr>
<td>V2</td>
<td>25</td>
<td>13.1</td>
<td>3.206</td>
<td>1.495</td>
<td></td>
<td>480.5796</td>
<td>8.7725</td>
<td>16.5016</td>
<td>0.10993</td>
</tr>
<tr>
<td>V3</td>
<td>35</td>
<td>11.5</td>
<td>2.8144</td>
<td>1.9707</td>
<td></td>
<td>444.9028</td>
<td>10.9564</td>
<td>19.7196</td>
<td>0.11992</td>
</tr>
<tr>
<td>V4</td>
<td>45</td>
<td>9.7</td>
<td>2.3739</td>
<td>2.3739</td>
<td></td>
<td>408.4496</td>
<td>12.2893</td>
<td>21.2408</td>
<td>0.13991</td>
</tr>
<tr>
<td>V5</td>
<td>55</td>
<td>7.6</td>
<td>2.6563</td>
<td>1.86</td>
<td></td>
<td>467.2571</td>
<td>12.1968</td>
<td>20.5049</td>
<td>0.16989</td>
</tr>
</tbody>
</table>

Table 4.2 Boundary conditions used for tornado simulation.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>where</th>
<th>B.C. Type</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABCD</td>
<td>fig. 4.2(b)</td>
<td>“velocity inlet”</td>
<td>to specify circulation by $v_\theta$ and mass flow rate by $v_r$</td>
</tr>
<tr>
<td>ABMN</td>
<td>fig. 4.2(b)</td>
<td>“viscous wall”</td>
<td>-</td>
</tr>
<tr>
<td>CDEF</td>
<td>fig. 4.2(b)</td>
<td>“viscous wall”</td>
<td>-</td>
</tr>
<tr>
<td>MNGH</td>
<td>fig. 4.2(b)</td>
<td>“inviscid wall”</td>
<td>It is difficult to specify as open, so inviscid wall is used.</td>
</tr>
<tr>
<td>CDIJ</td>
<td>fig. 4.2(a)</td>
<td>“viscous wall”</td>
<td>-</td>
</tr>
<tr>
<td>IJK</td>
<td>fig. 4.2(a)</td>
<td>“outflow”</td>
<td>interpolate the boundary values from interior</td>
</tr>
<tr>
<td>OGH</td>
<td>fig. 4.2(b)</td>
<td>“viscous wall”</td>
<td>-</td>
</tr>
<tr>
<td>BCKJ</td>
<td>fig. 4.2(a)</td>
<td>“rotational periodic”</td>
<td>set same as ADKI to satisfy axisymmetric condition</td>
</tr>
<tr>
<td>ADKI</td>
<td>fig. 4.2(a)</td>
<td>“rotational periodic”</td>
<td>set same as BCKJ to satisfy axisymmetric condition</td>
</tr>
</tbody>
</table>
4.2 Distributions of Key Parameters

4.2.1 Flow Field in the Outer Region

Since the flow structures of various cases were similar to each other, the analysis in this section was based on case V5. Other cases will be presented afterwards. The azimuthal velocities are shown in fig. 4.5 through fig. 4.9 for cases V1 through V5. They are normalized by their maximum values, which are listed in table 4.1. Each vertical distance \( z \) was normalized by each \( RMW \). These data were collected in each case up to elevation of 1.0 m. Because \( RMW \)s were different for each case, the normalized values have different scales. The dotted lines in these figures indicate zero values. So the curves left to the dotted lines correspond to negative values, and right correspond to positive values. This has specific graphical meaning with respect to the radial velocities \( v_r \). The curves on the left side of the dotted line indicate that the flow is coming in and the curves on the right side of dotted line indicate that the flow is coming out. The index \( i = 1, \ldots, 10 \) corresponds to values at radius of 1st to 10th \( RMW \). As can be seen in these \( v_\theta \) and \( v_r \) figures, there should be an outer region and inner region of whole domain of interest. The outer region is basically from radius of 10 \( RMW \)s to 2 \( RMW \)s. The inner region is from the center axis to radius of around 2 \( RMW \)s.

The variations of \( v_r \) and \( v_\theta \) in the outer region is relatively small in terms of elevation and radius in the vertical distances, especially in the higher elevation. In other words, the values are varying monotonically with respect to the vertical distances next to the floor and stays vertically constant at higher levels. This situation is also shown in the \( v_\theta/v_{\theta\text{max}} \) contours in fig. 4.11(a). For case V5, contour lines are pretty much vertical until down next to the floor. The magnitudes of the radial velocities \( v_r \) are apparently increasing with respect to decreasing radius. This means that the flow is accelerating as it is approaching the center of the domain especially next to the floor until the radial velocity reaches the minimum negative values around the \( RMW \). This acceleration is more apparent at lower elevation as can be seen in fig. 4.5(a) through fig. 4.16(a). However, the height of this acceleration in the radial direction decreases with respect to decreasing radius. As the flow approaches the center, the bulk flow of negative radial velocities \( v_r \) approaches the floor as well, with the magnitude of the negative \( v_r \) increasing and finally reaching their maximum values.
around \( RMW \). However, as seen here, closer to the center, the flow begins to move outward (in positive radius direction) above the bulk negative \( v_r \). This is illustrated in the profiles correspond to \( i = 1 \) in fig.4.5(a) through 4.16(a).

Taking into account the figures 4.5(b) through 4.16(b) of \( v_\theta \), the trend of the flow with respect to radius is clear. The circulation of the flow with respect to \( z \)-axis \( \Omega_z = v_\theta \cdot x \) or \( \Omega_{\text{normalized}} = \left( \frac{v_\theta}{v_{\theta\text{max}}} \right) \left( \frac{x}{RMW} \right) \) stays at a platform until radius of 2 \( RMWs \), as shown in fig.4.10(b). This is also shown in fig.4.5 through 4.9 in that since the circulation \( \Omega_z \) or \( \Omega_{\text{normalized}} \) is basically constant, the \( v_\theta/v_{\theta\text{max}} \) must be increasing with respect to decreasing \( x/RMW \). So in the outer region, the flow field can be summarized as follows

1. the flow is accelerating in the radial direction;
2. the flow is rotating more and more quickly in the circumferential direction as it is approaching the center;
3. up to the twice \( RMW \), the circulation with respect to the center \( \Omega_z \) was essentially constant.

4.2.2 Flow Field in the Inner Region

In the inner region, the flow field is much more complicated than in the outer region. The inner flow field in case V5 is shown in fig.4.11. The dashed lines with arrows in fig.4.11 denote the velocity vector in the meridian plane \( \vec{v}_{\text{meridian}} = (v_r, v_z) \) with longer dashes denoting greater magnitude \( |\vec{v}_{\text{meridian}}| = \sqrt{v_r^2 + v_z^2} \); the gray contours in fig.4.11(a) denote the normalized azimuthal velocity \( v_\theta/v_{\theta\text{max}} \) with increment of 0.05; the gray contours in fig.4.11(b) are the normalized T.K.E. \( k/v_{\theta\text{max}}^2 \) with the increment of 0.005. The darker the gray scale, the higher the values. Actually, because of the axisymmetric assumption carried out in this simulation, there should be no radial component \( v_r \) and azimuthal component \( v_\theta \) on the central axis \( (x, y, z) = (0, 0, z) \). However, this requirement was not 100\% fulfilled in this simulation as shown in fig.4.11 in that some streamlines were not tangential to the \( z \)-axis, which means \( v_r \) was not completely zero on the \( z \)-axis, though their values are extremely small. Another observation is that the magnitudes of normalized velocities were on the order of \( O \left( 10^{-1} \right) \) in the low speed region (where the dashed line spacing is very small), while the
normalized T.K.E. was essentially of the same order $\sqrt{\mathcal{O}(10^{-2})} \sim \mathcal{O}(10^{-1})$. This means that the turbulence intensity was around 100%. Because of the extraordinary high values of the turbulence intensity, the flow in the center must be highly unsteady. On the other hand, the flow in the central region is very slow, which indicate that the flow there may not be turbulent and highly unsteady as indicated by the turbulence intensity. This is a flat contradiction resulting from the assumptions based on which the simulations were carried out. If we consider the velocities in the simulation, we notice the velocity was somewhat “averaged”, because the velocities predicted by governing equation, boundary condition, computational domain and turbulence models were mean velocities, i.e. the governing equations are Reynolds Averaged N-S equations, the velocity boundary condition refers to the “Reynolds Averaged” velocity components. Similar process of averaging is used in processing experimental data to locate the center of the primary vortex in the tornado simulator. Many of Lewellen’s simulation [10] and analysis are based on the mean velocities. His snapshots at different instant shows that the primary vortex around the central region is also moving around the center periodically. This periodicity may result in the low values of the speed in central region and relatively higher values of T.K.E., which is actually a sum of square fluctuation velocities. The contradiction mentioned above is a obvious indication that the steady and axisymmetric assumption of this simulation was not physically reasonable. However, the analysis of the flow in the outer region should be worthwhile. And the analysis of the inner region, though not that credible, is also a very good reference.

First of all, the contours are consistent to the analysis made in the last section based on fig. 4.5 through fig. 4.9. The flow in the outer region that is coming into the inner region “collides” with the vortices illustrated by the rotating streamlines around $x = 1 \text{ RMW}$ and $y = 1.2 \text{ RMW}$. Then the incoming flow is “pushed” back in the radial direction. So both of the minimum and maximum radial velocity, $v_{r_{\text{min}}}$ and $v_{r_{\text{max}}}$, occur at the same radial location of about one $\text{RMW}$ as illustrated by the horizontal dashed streamlines.

Other than this, the flow is also accelerating in the azimuthal direction as it is approaching the $z$-axis, eventually encounters the central vortices, then the azimuthal velocity $v_{\theta}$ ceases to increase.
and reaches its maximum value. This mechanism is basically the same as \( v_r \) discussed above. So both the magnitudes of azimuthal velocity \( v_\theta \) and radial velocity \( v_r \) reach their maximum values around the same place with the radius of one \( RMW \), which is shown in fig. 4.11(a).

Besides the \( v_r \) and \( v_\theta \)’s variation, another structure should be important. In the lower level of the region of interest, there are “horizontal” vortices (HV) aligning parallel to the floor. The structure of these HVs is pretty complicated, though they are similar in terms of various cases. The function of this HV is to twist the flow from coming to the center almost horizontally to going vertically upward to the outlet of the domain. These vortices are enclosed by the incoming flow, eventually the incoming flow climbs over the HV and keeps going upward out of the outlet. So if these HVs are considered kind of bubbles around \( z \)-axis, though complicated, the reattachments take place at the \( z \)-axis at distance of 4 \( RMW \)s in case V5, as shown in fig. 4.11. The distributions of T.K.E. are interesting. The T.K.E.s reach extrema at the center of the HV, which means this extremum region coincides with the HV. At higher elevations of 3 \( RMW \), T.K.E. is increasing with respect to increasing vertical distances. The T.K.E.s are greater in the higher level because the speed in that region is relatively higher and relatively more non-axisymmetric than the flow in the lower levels. Figures 4.12 through 4.15 shows the normalized T.K.E. \( k/\theta_{\theta,\max}^2 \), \( v_\theta/v_{\theta,\max} \), static pressure \( p_{\text{static}}/\frac{1}{2}\rho \theta_{\theta,\max}^2 \) and \( \Omega_{\text{z,normalized}} = (v_\theta/v_{\theta,\max})(x/RMW) \).

### 4.3 Vortical Structures

The vortical structures of corner flow in the inner region were described in section 4.2.2. Though the structures in each of the meridian planes are supposed to be identical because of the construction of the model and the boundary conditions, they were not actually. Fig. 4.16 shows the iso-surfaces of normalized vorticity in \( z \) direction and normalized second characteristic invariant \( \Pi_2 \) of velocity gradient tensor, \( \overrightarrow{D} \), proposed by Miyahchi T. et al. [19].

\[
\overrightarrow{D} = \begin{pmatrix}
\frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\
\frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\
\frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z}
\end{pmatrix}, \tag{4.1}
\]
\[ \mathbf{A}_2 = \begin{vmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} \end{vmatrix} + \begin{vmatrix} \frac{\partial w}{\partial x} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial y} & \frac{\partial v}{\partial z} \end{vmatrix} + \begin{vmatrix} \frac{\partial u}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial w}{\partial y} \end{vmatrix}. \tag{4.2} \]

This rectangular domain is extracted from pie grids, with \( x/RMW \sim 4, \ y/RMW \sim 4 \) and \( z/RMW \sim 3 \). Because our data are in a \( \frac{\pi}{4} \) domain, Tecplot did an extrapolation in another \( \frac{\pi}{4} \) domain so that the two domain can combine to be a rectangular domain, inside which is easier to construct iso-surfaces. Unfortunately, this extrapolation caused the data in the \( \frac{\pi}{4} \) domain next to the original domain not cylindrical as they are physically supposed to be.

Though deviated from an exact axisymmetric assumption, the vortical structure still have some features closely related to axisymmetric assumption and descriptions in section 4.2.2. The vortices shown in the meridian plane in fig. 4.11 are aligned parallel to the floor around the \( z \)-axis. Basically, there are two kind of vortical structures in this field: double vortex rolls aligned immediately parallel to the floor and the funnel-like iso-surfaces above the double vortex rolls.

The evolution of the funnel-like iso-surfaces from the far field are interesting. As is shown in fig. 4.10(b), fig. 4.12(d) through fig. 4.15(d), the contours of circulation with respect to the \( z \)-axis are nearly perpendicular to the floor, which means the outflow is horizontally dominated. This feature is also illustrated in fig. 4.16 in that the iso-surfaces in the outer region are much more cylindrical than funnel-like. However, in the inner region, the shape of the iso-surfaces are more funnel-like than cylindrical. The change is due to the acceleration of the radial velocities at lower levels, which transports much more vorticity to the floor than the radial velocity on higher levels does. This mechanism causes the cylindrical iso-surfaces to twist more to the floor and become funnel-like. Another outcome of this mechanism is again the acceleration of the azimuthal velocities. Since the circulation with respect to \( z \)-axis or vorticity in the \( z \)-axis direction is transported much more to the floor than to the above, the azimuthal velocities reach their maximum values next to the floor, which are addressed in the section 4.2.2.

Another mechanism concerns the radial velocity. Since the flow is eventually going almost ver-
tically upward, the flow must be directed to the outlet above the central inner region. This means that the radial velocities in the horizontal plane must be twisted upward, resulting in a circulation perpendicular to all of the meridian planes. This circulation orthogonal to the meridian planes resulted in a circular vortex rolls aligned parallel to the floor around the $z$-axis. These double vortex rolls are shown by the iso-surfaces of $\Pi_2$ in fig. 4.16.

More than this, another vortex roll will be induced by this vortex roll. They form a double vortex roll aligned parallel to the floor around the $z$-axis. Since the radial velocities on higher levels are also supposed to be twisted upward, the same mechanism occurs at higher levels too, which causes not only the double vortex rolls, but also a central circulation region up to relatively higher levels of $z/RMW \sim 3$ in case V5.
Figure 4.1 A schematic illustration of tornado/microburst simulator.
(a) Some boundaries.

(b) Channel to direct the flow into the domain.

Figure 4.2 The computational domain used for tornado simulation.
Figure 4.3  The \( y_+ \)s against the radius for cases V1 through V5.

Figure 4.4  The azimuthal velocities \( v_\theta \) at the inlet \( ABCD \) and the maximum azimuthal velocities \( v_{\theta \text{max}} \).
Figure 4.5 The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$ for case V1.
Figure 4.6 The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$ for case V2.
Figure 4.7 The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$ for case V3.
Figure 4.8 The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$ for case V4.
Figure 4.9 The normalized numerical and experimental velocity components, $v_r$ and $v_\theta$ for case V5.
(a) Normalized Static Pressure $p_{\text{static}}/\frac{1}{2}\rho v^2_{\theta_{\text{max}}}$.

(b) $\Omega_z \text{ normalized} = (v_{\theta}/v_{\theta_{\text{max}}}) (x/\text{RMW})$.

Figure 4.10 The static pressure and normalized circulation $\Omega_z$ in case V5.
Figure 4.11 The normalized $v_θ$ and T.K.E. in case V5.
Figure 4.12  The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V1.
Figure 4.13 The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V2.
Figure 4.14 The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V3.
Figure 4.15  The normalized $v_\theta$, T.K.E., $p_{\text{static}}$ and $\Omega_z$ in case V4.
Figure 4.16 The normalized $\omega_z$ and $\Pi_2$ in case V5.
CHAPTER 5. Summary and Conclusions

For this thesis, the numerical simulation of straight line flow and tornado-like vortex flow has been carried out. In the numerical simulation, “standard wall function”s, which do not require grids to be fine next to the floor, do not perform correctly with respect to the experimental results and law of wall. However, the “enhanced wall function” available in FLUENT, which incorporate both the $k-\ell$ and $k-\varepsilon$ models and requires much finer grids next to the floor combined with riblets on the floor to imitate the chain roughnesses, show improved performance with respect to the experimental results and law of wall. The mean velocity profile $u(z)$ and boundary layer thickness $\delta$ predicted by the “enhanced wall function” agree very well with experimental counterparts. The normalized mean velocity profile in terms of wall units $\ell_+ = \nu/u_*$ matches reasonably well with the law of wall (linear layer $u_+ \sim y_+$, log layer $u_+ \sim \log y_+$ and buffer layer).

However, another important parameter, the turbulent kinetic energy (T.K.E.), $k$ was not correctly predicted until the Reynolds stress Model (RSM) was employed. The predicted T.K.E. by RSM was much better than those by “enhanced wall function” because each of the Reynolds stresses, especially the predicted diagonal fluctuation velocity correlations $\overline{u'u'}$, $\overline{v'v'}$ and $\overline{w'w'}$ behave almost correctly, according to the data by David B. DeGraaff and John K. Eaton [1] shown in fig. 1.3.

Other the parameter distribution at the outlet of the computational domain or the inlet of the test section, the numerical flow field next to the riblets were studied. The first riblet contributes most to the turbulent boundary layer development. Although a “steady” solver was used in this simulation, some unsteady characteristics were observed instead. These characteristics refer mainly to the apparent shedding vortex stemming from the first riblet roughness, which were identified by the discriminant $\lambda_2$ by Adrain [15]. However, as the boundary layer develops, this unsteadiness
was not observed near the following riblets. As the roughnesses were sparsely distributed on the floor, each one was followed by a couple of recirculating regions or bubbles (primary bubble and secondary bubbles) next to the floor. Almost all the primary bubbles had the same dimension and structure. In front of the first riblet, no bubbles were observed, but there were bubbles observed from the second to the seventeenth bubbles. Details of these bubbles were illustrated in chapter 3.

The cases for the tornado simulation that have been run corresponds to those by Haan et al. [3]. However, the numerical corner flow structures were different than those observed in tornado/microburst simulator. All of numerical structures were so called “two-celled structure”. The variations of radial velocity and tangential velocity were studied in terms of decreasing radius in the surface layer. The boundary layers were also described qualitatively from far field to the central region. The vortical structures, from the far field to the central axis, were studied too in terms of the velocity field. Basically two kind of vortical structures were observed and described. One of them is the cylindrical vortex tube, which involved a funnel like vortex in terms of decreasing radius. Another important vortical structure is the double vortex rolls that aligned parallel to the floor about the central axis. These vortex rolls results from the rapid change of the flow from radial direction to vertical direction. These variation corresponds to not only the double vortex rolls but also the rapid change of azimuthal velocity and radial velocity, and in turn results in the maximum azimuthal velocity $v_{\theta_{\text{max}}}$ and maximum radial velocity $v_{r_{\text{max}}}$. 
BIBLIOGRAPHY


