A study on air flow and odor emission rate from a simplified manure storage tank

Qianbao Liu
Iowa State University

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

Part of the Agriculture Commons, Bioresource and Agricultural Engineering Commons, and the Mechanical Engineering Commons

Recommended Citation
Liu, Qianbao, "A study on air flow and odor emission rate from a simplified manure storage tank" (1994). Retrospective Theses and Dissertations. 11287.
https://lib.dr.iastate.edu/rtd/11287

This Dissertation 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 Retrospective Theses and Dissertations by an authorized administrator of Iowa State University Digital Repository. For more information, please contact digirep@iastate.edu.
INFORMATION TO USERS

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

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

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

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

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.
A study on air flow and odor emission rate from a simplified manure storage tank

Liu, Qianbao, Ph.D.

Iowa State University, 1994
A study on air flow and odor emission rate from a simplified manure storage tank

By

Qianbao Liu

A Dissertation Submitted to the
Graduate Faculty in Partial Fulfillment of the
Requirements for the Degree of
DOCTOR OF PHILOSOPHY

Department: Agricultural and Biosystems Engineering
Major: Agricultural Engineering

Approved:
Signature was redacted for privacy.

In Charge of Major Work
Signature was redacted for privacy.

For the Major Department
Signature was redacted for privacy.

For the Graduate College

Iowa State University
Ames, Iowa

1994
To my wife Lirong and son Bill
TABLE OF CONTENTS

LIST OF FIGURES ........................................................................................................ vi
LIST OF TABLES ........................................................................................................... viii
NOMENCLATURE .......................................................................................................... ix
ABSTRACT .................................................................................................................. xi

CHAPTER 1. INTRODUCTION ..................................................................................... 1

CHAPTER 2. OBJECTIVES .......................................................................................... 4

CHAPTER 3. LITERATURE REVIEW ........................................................................... 5
  3.1. Odor and Its Concentration Measurement .......................................................... 5
  3.2. Odor Emission Rate and Related Study ............................................................... 6
  3.3. Studies on Atmospheric Boundary Layer and Similitude ................................. 7
  3.4. Experimental Studies on Separation Flow ......................................................... 11
  3.5. Numerical Simulation of Separation Flow ......................................................... 14
  3.6. The Numerical Algorithms ............................................................................... 21
  3.7. Turbulence Modeling ......................................................................................... 24
    3.7.1. Classification of turbulence models .............................................................. 24
    3.7.2. The standard k-ε model and its limitations .................................................. 26
    3.7.3. Proposed modifications to the standard k-ε model ....................................... 27
      3.7.3.1. Low Reynolds number models ............................................................... 27
      3.7.3.2. The approach of Chieng and Launder .................................................... 30
      3.7.3.3. Replacing ε equation ............................................................................. 34
      3.7.3.4. Two layer approach ............................................................................. 36
      3.7.3.5. Incorporating Reynolds stress terms ..................................................... 39

CHAPTER 4. PROBLEM DEFINITION AND SIMPLIFICATION .............................. 41
  4.1. Problem Description ........................................................................................ 41
  4.2. Selection of Method ......................................................................................... 41
    4.2.1. An open bottom wind tunnel floating on manure ........................................ 41
    4.2.2. A box covering the tank ............................................................................. 42
    4.2.3. Similitude study ........................................................................................ 43
    4.2.4. Numerical simulation ............................................................................... 45
    4.2.5. Method selected ........................................................................................ 46
  4.3. The Assumptions ............................................................................................. 47
  4.4. Simplification from a Three-Dimensional to a Two-Dimensional Problem ....... 47
CHAPTER 5. MATHEMATICAL MODEL

5.1. The Time Averaged Governing Equations

5.1.1. The equations for laminar flow

5.1.2. The time averaged governing equations

5.2. Selection of Turbulence Model

5.2.1 Model selection

5.2.2. Rodi's two layer turbulence model

5.2.2.1. The standard k-ε model

5.2.2.2. The one equation near-wall turbulence model of Rodi

5.2.2.3. The two layer model

5.3. The Diffusion Coefficients

5.4. The Generalized Partial Differential Equations

5.5. Non-Dimensionalization of the Equations

5.6. Boundary Conditions

CHAPTER 6. COMPUTATIONAL PROCEDURE

6.1. Treatment of the Source Term

6.2. The Discretized Equation

6.3. The Solution Procedure

6.4. The Relaxing Coefficients

6.5. Convergence Criteria

6.6. Grid Arrangement

6.7. Emission Rate Calculation

6.7.1. Emission rate calculation for the 2-D tank segment

6.7.2. Emission flux calculation for the 2-D tank segments

6.7.3. Emission flux for the tank

6.7.4. Wind speed correction

6.7. Computer Equipment

CHAPTER 7. EXPERIMENTAL ARRANGEMENT

7.1. Wind Tunnel Setup

7.2. The Flow Field Entering the Test Section

7.3. The Scaled Model of the Simplified Tank Segments

7.4. The Flow Pattern Visualization and Air velocity measurement

CHAPTER 8. RESULTS AND DISCUSSION

8.1. The Verification of the Numerical Prediction

8.1.1. The flow pattern verification

8.1.1.1. The case of tank full

8.1.1.2. The case of half full tank

8.1.1.3. The case of empty tank
vi

LIST OF FIGURES

Figure 1. An open manure storage tank .................................................................................. 3
Figure 2. The two layer and three layer treatments ................................................................. 31
Figure 3. Dividing the tank into rectangles ............................................................................ 49
Figure 4. The computation domain after simplification ......................................................... 50
Figure 5. Typical control volumes .......................................................................................... 69
Figure 6. Dividing the computation domain into sections ..................................................... 76
Figure 7. The grid arrangements .............................................................................................. 77
Figure 8. A typical section for emission rate calculation ......................................................... 80
Figure 9. Wind tunnel setup ..................................................................................................... 84
Figure 10. Points where the velocity was measured ............................................................... 86
Figure 11. The velocity profile along line AB in Figure 10 ..................................................... 86
Figure 12. A light box for flow visualization .......................................................................... 89
Figure 13. The predicted and experimental flow pattern for a full tank (W/H=2, h/H=1) .... 93
Figure 14. The predicted and experimental flow pattern for a half full tank ......................... 94
Figure 15. The predicted and experimental flow pattern for an empty tank (W/H=2, h/H=0) 95
Figure 16. The air velocities along the center of the tank (W/H=2, h/H=0) ......................... 97
Figure 17. The flow pattern in the tank region (W/H=2, h/H=0.5) ...................................... 99
Figure 18. The velocity contour in the tank region (W/H=2, h/H=0.5) .............................. 100
Figure 19. The odor concentration contour in the tank region (W/H=2, h/H=0.5) ............. 101
Figure 20. The flow pattern in the entire domain (W/H=2, h/H=0.5) ............................................102
Figure 21. The odor concentration contour in the entire domain (W/H=2, h/H=0.5) .............103
Figure 22. The air flow pattern at Re=3x10^5 (W/H=2, h/H=0.5) ...........................................108
Figure 23. The air velocity contour at Re=3x10^5 (W/H=2, h/H=0.5) ..............................109
Figure 24. The air flow pattern at Re=1.4x10^4 (W/H=2, h/H=0.5) .................................110
Figure 25. The velocity contour at Re=1.4x10^4 (W/H=2, h/H=0.5) ..................................111
LIST OF TABLES

Table 1. The standard k-ε model constants ..............................................56
Table 2. Summary of equations .................................................................62
Table 3. The constants used in the equations ...........................................62
Table 4. The boundary conditions ...........................................................65
Table 5. The treatment of the source terms ..............................................67
Table 6. The relaxing coefficients ............................................................73
Table 7. The grid arrangements ...............................................................75
Table 8. The predicted recirculation lengths ..........................................104
Table 9. Calculated Odor fluxes .............................................................105
Table 10. Odor emission fluxes at different Reynolds numbers ................112
Symbols:
A: Area (m²).
B: Constant in the wall function (= 5.0)
c: Odor concentration (ou)
c₁, c₂, c₃, c₄: Turbulence model constants.
D₀: Odor molecular diffusion coefficient (m²/s).
D: Tank diameter (m).
Fr: Froude number.
G: Turbulence generation.
h: Manure depth in the tank (m).
k: Turbulence energy (m²/s²).
L, l: Length scale (m).
q: Odor emission rate from a control volume (ou*m/s).
Q: Odor mission rate from the tank (ou*m/s).
T_r: Residual of the continuity equation.
R_y: Turbulence Reynolds number, k½y/v.
s: The source term in the generalized equation. See Equation 5.22
Sc: Schmidt number.
u, v: Velocity components.
u₀: Free stream velocity or velocity at the top of the computation domain.
v_w: Wind velocity measured at 10 m from the ground (m/s).
v⁺: Dimensionless velocity as defined by friction velocity.
v*: Friction velocity, (τ_w/p)½
x: Coordinate direction.
y: Coordinate direction or distance to solid surface
y*: Dimensionless distance from the solid surface, k½y/v or dimensionless y.
y⁺: Dimensionless distance as defined by friction velocity. See Equation 5.16.
y_min: Minimum distance to any solid surfaces.
z: Distance above ground (m).
e: Isotropic turbulence dissipation rate (m²/s³).
ϕ: Generalized variable.
μ: Absolute viscosity (kg/m/s).
v: Kinematic viscosity (m²/s).
σ: Turbulent Prandtl or Schmidt number.
Γ: Generalized diffusion coefficient.
ρ: Density.
δ: Boundary layer thickness (m).
\( \delta_{ij} \): Kronecker delta.

\( \omega \): The relaxing coefficient or as defined in Equation 3.7.

\( \kappa \): Von Karman's constant (= 0.4).

\( \tau_w \): Shear stress at the wall (kg/m/s^2).

Subscripts:

eff: Effective.

e, w, n, s: Location as in relation to the grid point. See Figure 5.

E, W, N, S: Location as in relation to the grid point. See Figure 5.

i, j: Grid point notation.

nb: Neighboring points

t: Turbulent.

Superscripts

*: Non-dimensionlized variable.

\( \prime \): Fluctuating component or correction term.

\( \bar{} \): Time averaged component.

+: Dimensionless turbulent parameter.

n, n-1, n+1: Iteration number.
This study presents a numerical and experimental evaluation of air flow and odor emission rate from an open manure storage tank. The concentration at the manure surface, the tank dimensions, and wind speed were used to calculate air flow and the emission rate.

The three-dimensional air flow in the open tank was divided into four segments and each segment was treated as two-dimensional. The SIMPLER algorithm developed by Patankar and a two layer turbulence model were used in the numerical simulation with a grid of 152x139.

The predicted emission rate agreed with the field measurement results found in the literature. Experimental verification of the air flow showed that the flow pattern and velocity profile prediction were also in agreement with the experimental results.

The calculated odor emission rate was a function of the manure surface area, the odor concentration at the manure surface, tank dimensions, and the wind speed. Reynolds number sensitivity was tested by running the simulations for a wide range of Reynolds numbers.
CHAPTER 1. INTRODUCTION

Odor is one of the major environmental concerns for the livestock industry. The manure storage facility is one of the major odor sources. An adequate estimation of the odor emission rate from a manure storage facility is needed to facilitate an odor dispersion model, to help compare different systems, and to reduce the odor emission by analyzing how different factors affect the odor emission rate.

Currently, adequate estimation of the emission rate from a manure storage tank or lagoon is not available. The odor emission from a production facility can be estimated by knowing the exhaust air flow rate and odor concentration in the exhaust. For a manure storage facility, the problem is more complicated because the air exchange rate is difficult to obtain. Li (1994) tried to back calculate the odor emission rate from field measurements of an odor plume width and odor intensity downwind using the Gaussian plume model. Bode (1991) studied odor and ammonia emission by covering the tanks. A fan was provided to ventilate the tank. But the result is difficult to be related to wind speed or different sizes of tanks. Carney and Dodd (1989) calculated the emission rate from manure storage or treatment facilities by multiplying the odor concentration at the source by surface area of the source and by the prevailing wind speed. How the concentration at the source was measured was not mentioned. Li (1994) showed that the approach of Carney and Dodd greatly over estimated the emission rate if the concentration was taken at the manure surface.
Two major components affect the odor emission rate from a manure storage or treatment facility: 1) The state of the manure and the bio-process taking place that controls the production. If there is no production, there would be no emission. 2) The state of the air above the manure which controls how fast the odor can be transported. If the tank is sealed off, there would also be no emission. The two components affect each other. A complete modeling of odor emission process needs to consider the two components. Currently, however, the bio-processes in manure has only been linked to some of the odor contributing chemicals but not odor itself (Zhang, 1992).

Modeling the odor production inside the manure with adequate accuracy has not been studied. This is in part due to the lack of understanding about odors and the bio-process that generates odor. Even if it is possible, it might not be of practical use in dispersion modeling because of the numerous variables required to predict the process. This study estimates the odor emission rate by focusing on the air above the manure. Given the wind speed and the tank dimensions, the air flow in the tank can be predicted. Given the concentration of odor at the manure surface or the odor concentration in the air that is infinitely close to the manure surface, the emission rate can be predicted. The bio-process in the manure is counted for by the concentration at the manure surface. Different states of manure would have different odor concentrations at the manure surface giving the same air flow above the manure. Using this approach, the emission rate can be estimated by measuring the odor concentration at the manure surface and given the dimensions of the tank and wind speed. This approach is practical and can also apply to lagoons and other similar odor sources. After comparing
different methods to predict the flow in the tank, numerical simulation was selected for this study.

The tank of interest was an open round tank sitting on flat ground as shown in Figure 1. It has a diameter of $D$ and a height of $H$. $h$ is the manure depth and $u_0$ is the wind speed.

Figure 1. An open manure storage tank
CHAPTER 2. OBJECTIVES

The purpose of this study is to determine the odor emission rate from the manure storage tank shown in Figure 1. Specifically, the objectives of this study were to:

1. determine the odor emission rate from an open manure storage tank given the manure surface odor concentration, wind speed, and tank dimensions,
2. compare numerical simulation with other possible methods,
3. identify the appropriate numerical scheme and turbulence model to be used for the study,
4. experimentally verify the numerical simulation, and
5. find out the usability of the simulated results under conditions that are different from the simulated condition.
CHAPTER 3. LITERATURE REVIEW

3.1. Odor and Its Concentration Measurement

Odor is the sensation caused by odorant acting on the sense of smell. Quantitative odor concentration measurement is basically a threshold measurement. It measures quantitatively how many times more concentrated the odor is than its threshold concentration. The measurement is made using an olfactometer and a panel of human subjects. Various variations of olfactometer have been developed. Research on the current dynamic dilution olfactometers started in the United States in the middle to later 70's by Dravnieks and his co-workers (1975,1978), which led to American Society of Testing Material (ASTM) standard E679: Determination of odor and taste by a forced-choice ascending concentration series methods of limits (1989). A detailed review of research related to odor, odor measurement, and odor control was done by the committee on odors from stationary and mobile sources (1979). In recent years, olfactometers have been developed and used for agricultural related use in European countries and in Australia. Carney and Dodd (1989a), Jones et al. (1992), and Hangartner (1989) discussed in some detail about olfactometer design and recommended parameters used in their designs. An olfactometer has been developed and used at Iowa State University by Bundy and Liu (1993).

Odor concentration is expressed in terms of odor unit (ou), or how many times more concentrated than the threshold concentration.
3.2. Odor Emission Rate and Related Study

Odor is a nuisance only when it affects others on their property or public property. Some of the recent research on odor were to find out how far down wind the odor can be detected given the odor source and other conditions. For these studies, the odor emission rate from the source is necessary. Li et al. (1994) set up a grid system around a swine manure storage tank to measure the odor intensities and odor plume width from the tank. The data were analyzed to estimate the emission rate. Li estimated the emission rate based on the plume width at a given distance from the source using the Gaussian plume model.

Carney and Dodd (1989b) used a Gaussian plume model to predict odor dispersion. Experiments were conducted to verify the model. For the open manure storage tank used, the odor emission rate was calculated as the product of the odor concentration at the source, the surface area of the tank, and the prevailing wind speed. The study found good agreement between the experimental results and predicted results. The location of odor concentration measurements were not given. Other research on dispersion studies (Smith, 1993; Gassman, 1992) did not discuss the issue of odor emission rate.

Bode (1991) did an experimental emission rate study of ammonia and odor in small slurry tanks of 2 m high and 1.9 m in diameter. The slurry was stored for a 5 month period. During that period, the odor and ammonia emissions were measured. His study used a cover over the tank with a fan to provide air exchange with the outside. The fan provided an air flow rate of 48 m³/min at 1 m/s. The odor concentrations of 120-200 ou were reported.
Open bottom wind tunnel type of devices were often used to estimate the odor emission rate from spreading manure. The device used was a wind tunnel without the bottom. Fresh air was pulled through the device and the exhaust measured to calculate the emission rate (Homans, 1988).

Zhang (1992) studied degradation of swine manure and ammonia emission from manure pits. Degradation of swine manure under pit was simulated in the laboratory. The chemical and biological processes in the manure were reported to be different in different layers. Variation of the manure characteristics and ammonia generation rate in each layer was determined. It was found that the ammonia generation rate within the manure varied in an inverse proportion with storage time. Four level of air velocity above the manure (0 m/s, 0.203 m/s, 0.406 m/s, and 0.508 m/s) were used to study the impact of air velocities on the ammonia release rate. It was found that for a given surface concentration of ammonia in the manure, higher velocity resulted in higher release rate. Based on the experimental results, a computer model was developed to predict the ammonia release rates under different conditions.

3.3. Studies on Atmospheric Boundary Layer and Similitude

Much research has been done to study the atmospheric flow characters, the flow around structures, and pollution dispersion in the atmosphere. Because of the different characteristics in the atmosphere, the atmosphere is divided into different layers (Stull, 1991). The atmospheric boundary layer (ABL) is part of the atmosphere that is directly influenced
by the presence of the earth's surface, and responds to surface forces with a time scale of approximately an hour or less. The ABL can be from several hundred meters to 1 km in depth (Stull, 1991). Most of the human activities happen in the ABL. The atmospheric surface layer (ASL) is the layer about 100 m in height from the earth's surface (Cermak, 1992). It is part of the ABL. Low rise buildings and dispersion of air pollutants from low-level sources are in this layer.

Chok (1988) measured mean and turbulence properties in the ASL. The mean wind velocity profile in the vertical direction was reported as,

$$\frac{u(z)}{u(10)} = \left(\frac{z}{10}\right)^{0.14}$$

where,

- $u = \text{the wind speed}$
- $z = \text{the height above the ground}$

Equation 3.1 is in agreement with other reported profiles (Stern, 1984). The measured intensity of the turbulence component, $\sqrt{\frac{\langle u'^2 \rangle}{u(z)}}$ was about 15%.

Similitude study in the ABL or the ASL needs a special wind tunnel to simulate the air flow condition in the atmosphere. Cermak (1981) studied wind tunnel design for the ABL flow simulation. He concluded that wind tunnels can be designed to physically model the ABL characteristics that are of primary significance. General design criteria for a wind tunnel that can physically model a wind range of the ABL conditions were given. Flow conditioning devices may be employed to create mean velocity distributions that resemble
distributions for turbulent boundary layers in the ABL. For the ASL, Cermak (1992) showed special flow conditioning is needed and can be achieved based on the ABL wind tunnels.

Snyder (1985) reviewed various aspects of similitude studies of pollutant transport and diffusion. On the similarity criteria, the square of the Froude number was pointed out as the most important parameter. The Froude number can be expressed as,

$$Fr = \frac{u_w}{\sqrt{\frac{gH}{\Delta \rho}}}$$

(3.2)

Fr^2 represents the ratio of inertial forces to buoyancy forces. The Rossby number, Ro, needs to be considered when modeling prototype flows with length scales greater than 5 km. The Rossby number represents the ratio of advective or local accelerations to Coriolis acceleration. The Reynolds number, Peclet number, and Schmidt number were generally dismissed as not important. It is assumed that at a high enough Reynolds number, the main structure of the turbulence will be totally responsible for the transport. It is also impossible for most cases to maintain the same Reynolds number for wind tunnel study. However, he pointed out that the effects of Reynolds number needed more study. Detailed discussion on similarity discussion was also given by Vermeulen (1980) and Snyder (1972).

Similitude studies in ABL or ASL wind tunnel were widely used to study the impact of wind on buildings and pollutant dispersion. Snyder (1994, 1990, 1985) did wind tunnel studies on air pollutant dispersion. Snyder (1994) conducted a wind-tunnel study to examine the effects of several parameters on the downwash of effluents released from stacks in the vicinity of rectangular-shaped buildings. The parameters included stack height, location,
wind direction, effluent-speed, etc. The measurements were to provide a better understanding of pollutant transport and provide data to evaluate the accuracy of numerical modeling. Snyder (1985) showed that field and laboratory observation of a concentration pattern on a hill surface under strongly stratified conditions showed very good agreement. The boundary conditions, impacts of terrain, and other aspects of similitude study were also discussed.

Papesch (1992) used an ABL wind tunnel to study the barrier spacing and porosity to reduce wind damage of kiwi fruit. Velocity profiles of

\[ \frac{U_{\epsilon+}}{U_{(10)}} = \left( \frac{x}{10} \right)^{0.2} \]

was adopted in the wind tunnel based on field measurement. Turbulence intensities from 10% to 30% were reported depending on the wind direction and ground roughness. Based on the result from this study, a barrier spacing to barrier height ratio of 4 was recommended to avoid vortices below half of the barrier height. A wind tunnel study carried out by Huber (1991) on building wake dispersions showed that the existence of a building had an influence on the dispersion near the building. The exact distance of influence was a function of building size. A modified Gaussian plume model was shown to be adequate to predict the dispersion.
3.4. Experimental Studies on Separation Flow

The flow around and inside the tank is a separation flow. Separation flows are widely encountered and studied. Because of the complexity of separation flow, a detailed study of flow structure and modeling of separation flow is a topic of great interest to researchers in fluid mechanics.

Simpson (1989) summarized the physical behavior of two-dimensional turbulent separated flows. He defined the separation as the entire process of departure, breakaway, or the breakdown of boundary-layer flow and significant normal to wall velocity component. For steady free stream separating turbulent boundary layers, the detachment state near the wall was proposed based on the fraction of time that the flow moves downstream. Incipient detachment (ID) occurs with instantaneous back flow 1% of the time. Intermittent transitory detachment (ITD) occurs with instantaneous back flow 20% of the time. Transitory detachment (TD) occurs with instantaneous backflow 50% of the time. Detachment occurs where the time averaged wall shearing stress is 0. The discussions were mainly on turbulence structure. He concluded that the wall function can be used very close to the detachment point. Simpson also pointed out the unsteadiness of separation flow. Effects of surface curvature on separation flow, flow over sharp-edged bluff bodies like backward-facing step and forward-facing step were also discussed. The unsteadiness of separated flow was shown in the review of flow over blunt bodies by Oertel (1990).

Bradshaw (1987) reviewed turbulent secondary flows. He distinguished two types of mechanisms of vorticity generation in secondary flow: “skew-induced” and “stress-induced”.
In "skew-induced" vorticity generation, vorticity is generated by quasi-invincid deflection of existing mean vorticity. It is found in both laminar and turbulent flow. Secondary flow generated on the surface with curvature is an example. In "stress-induced" vorticity generation, vorticity is generated by turbulent (Reynolds) stress, like the secondary flow generated in the corner of a square straight duct. The discussion emphasized the inadequate understanding of the problem, especially on the turbulence modeling of the turbulent stresses. He pointed out that only a few experiments on three dimensional flows included detailed measurements of all components of Reynolds stress. He concluded that basic physical understanding of the effect of mean-flow three dimensionality on turbulence structure is still lacking. The Reynolds stress turbulence model based on the understanding of turbulence structure also needs improvement. He concluded that engineering calculations will have to be done by Reynolds-averaged methods for the foreseeable future.

Schofield (1990) reviewed and analyzed the results from flow over surface mounted obstacles for the factors affecting the flow. For a single two-dimensional obstacle, the obstacle's impact on the pressure field depends on the configuration. The reattachment length ranged from 7-14 times the height of the obstacle. The wall function equation,

\[ u^* = \frac{1}{\kappa} \log y^* + A \]

where,

\[ u^* = \text{dimensionless velocity} \]

\[ y^* = \text{dimensionless distance from the wall} \]
\[ \kappa = 0.41 \]
\[ A = 5.0 \]

was reported to be valid for separation flow close to the separation point. Three dimensional obstacle was said to have more complex flow structure both upstream and downstream of the obstacle. The length of separated flow behind a three-dimensional obstacle was much smaller than a two-dimensional obstacle because a large proportion of the upstream flow goes around a three-dimensional object rather than over it. The reattachment can occur on top of a cube for a three-dimensional obstacle. Flow over multiple objects was also discussed.

Djilali (1991) experimentally studied the separation reattaching flow around a long rectangular plate placed at zero incidence in a low-turbulence stream. The study showed that the separated shear layer appears to behave like a conventional mixing layer over the first half of the separation bubble, where it exhibits an approximate constant growth rate. The characteristics of the shear layer in the second half of the bubble radically altered by the unsteady reattachment process. The length of the separation bubble was reported as 4.7 times of the plate thickness.

Hillier (1981) studied the free stream turbulence on the separation bubble. He reported a separation bubble of 2.72 to 4.88 times the plate thickness depending on the free stream turbulence. The higher the free stream turbulence, the shorter the bubble. The study also showed that the flow was essentially Reynolds number independent for \( \text{Re} \geq 2.7 \times 10^4 \).
Okamato (1992) studied the flow of surface mounted circular cylinders. It was found that
the H/D value had a strong influence on the separation size and drag coefficient on the
cylinder. The drag coefficient increases as the value of H/D increases. That was especially
true for H/D greater than 4. The recirculation region behind the circular cylinder enlarged
with an increase in H/D. The length of the recirculation region near the ground plane reached
maximum near H/D=4.

Three dimensional backward facing step flow was studied by Shih (1994). He found a
significant difference from two dimensional back facing step flow. Eckerk (1991) did an
experimental study on three-dimensional separated flow in front of a cylinder. A large-scale
fully developed vortex was formed in the plane of symmetry for low free stream velocity. It
was not present at a high free stream velocity.

The near wall behavior of two separated and reattaching flows formed by a sudden
expansion in a pipe was studied experimentally by Davenport (1991). It was found the near
wall flow in the separated flow was very different from a normal attached turbulent boundary
layer flow. Mean velocity profiles do not obey the wall function and can not be correlated
outside of the linear sublayer using the friction velocity. However, they do contain semi-
logarithmic regions.

3.5. Numerical Simulation of Separation Flow

Numerical simulation is widely used to simulate fluid flow. A perspective on the status
of current computational fluid dynamics (CFD) and future research directions were given by
Douglass and Ramshaw (1994) from the application point of view. They pointed out that the current state of CFD has yet to reach its full promise as a general tool for engineering design and simulation. SIMPLE (Patankar, 1980) type of algorithms are the most popular ones being used for engineering applications. Currently, considerable training in CFD is necessary to use CFD. The robustness, flexibility for complex flows, numerical errors and resolution were proposed as directions for further research. Boris (1989) reviewed the new developments in CFD area from new algorithms point of view. He pointed out that the CFD solution can be comparable to, or even exceed, the accuracy and resolution of laboratory experiments. Various new algorithms and novel schemes were presented.

Separation flow is widely encountered in the practical applications. Numerical studies of separation flow are topics of current interest. Majumdar and Rodi (1989) did a three dimensional numerical study of flow past a cylindrical structure on ground and simplified cooling towers. For the cylindrical structure on the ground, an orthogonal polar grid and standard k-ε turbulence model were used. The simulated cylinder had a height to diameter ratio of 1.9. SIMPLEC (Vandoormal and Raithby, 1984) scheme was used. A non-uniform grid of 45x42x40 was used with $y^+ \leq 150$ for grid points on the vertical cylinder surface and $y^+ \leq 299$ for ground and top of the cylinder. The inlet velocity profile used was,

$$u_m(z) = \begin{cases} 
  u_0 \left( \frac{z}{\delta} \right)^0.185 & z < \delta \\
  u_0 & z \geq \delta 
\end{cases}$$  

where,
The inlet kinetic energy, $k$, and turbulence dissipation, $\varepsilon$, were given based on measurement and were given as three groups: free stream, boundary layer and the point adjacent to solid surface. An experimental study was also done on a boundary layer wind tunnel. The results showed that the numerical simulation was capable of reproducing many of the complex flow features in the vicinity of cylindrical structure. It was found that the flow predicted was much more in a radical direction than in the experimental observation. The polar grid, which contributed to false diffusion for the problem, and not enough grid points were said to be the causes of the problem.

Paterson and Apelt (1989) numerically simulated the wind flow around a three-dimensional building. Cartesian coordinates and standard $k$-$\varepsilon$ turbulence model and SIMPLE scheme were used. The results were compared with experimental results reported in the literature. The agreement was reported to be comparable a well controlled wind tunnel test.

Murakami and Mochida (1989) numerically simulated the flow around a cube and a group of buildings. The standard $k$-$\varepsilon$ turbulence model was used. For the boundary conditions, the inlet velocity and turbulent kinetic energy values were measured in the wind tunnel. The turbulent dissipation for the inlet was

$$\varepsilon(z) = C_{\mu}k(z)^{3/2}/l(z) \quad 3.6$$

where,

$$z = \text{distance from the solid surface}$$
\[ l(z) = \text{length scale} \]
\[ c_\mu = \text{constant, } 0.09 \]

Four meshes and two types of solid boundary conditions were used. The densest one was a 50x49x28 non-uniform grid. One type of solid boundary condition was the tangential velocity profile was assumed to obey power law (1/7 power) and \( \varepsilon = c_\mu k^{3/2}/l \) with \( l = 1/2 \ c_\mu^{1/4} k y_{\min} \). The second type was the generalized boundary condition described by Launder and Spalding (1974). It was found that the grid resolution at the windward and leeward corner was very important. The boundary condition for \( \varepsilon \) at the solid wall has a significantly large influence on the flow field in the separation above the roof and also on the flow near the side walls at windward corners. The second type of boundary condition was found to be better. The numerical simulation of the k-\( \varepsilon \) two equation model with fine mesh can reproduce the mean velocity field and the mean pressure field around the model accurately. But there exist significant differences in the distribution of the turbulent energy around the windward corner and in the wake.

Kot (1989) reviewed some numerical schemes that were or could be used for numerical modeling of contaminant dispersion around buildings. He concluded that the k-\( \varepsilon \) model was the most widely used. Other models, namely different versions of Reynolds stress models, may not be feasible. The numerical methods suffer from the requirements of assuming values for many parameters.

Rizzetta (1994) numerically simulated the turbulent juncture flow field of a cylinder on a flat plate at high speed. A compressible version of low Reynolds number k-\( \varepsilon \) turbulence
model derived from Jones and Launder (1972) was used. The Beam-warming scheme was used to solve the partial differential equations. Non-uniform grids of 96x71x41, 144x106x61, and 191x141x81 were used to achieve a grid independent solution. The $y^+$ value for the subsonic flow was 1.85. The results showed all commonly observed physical features of the flow fields. The numerical result reproduced the experimentally generated surface streamline pattern, pressure distribution, and velocity field. The major deficiency was said to be the grid resolution in some parts of the domain and the inability to correctly predict the vortex strength and location. The k-ε model was able to provide a practical means for simulation of flow past complicated geometry.

Solberg and Eidsvik (1989) numerically simulated flow over a cylinder at a plane boundary. The standard k-ε model with a modified length scale was used. The turbulent length scale, $l_e = c_\mu \frac{3}{k^{3/2}} \epsilon^{-1}$ was modified according to one equation model. An orthogonal grid based on the velocity potential and stream function for a frictionless flow were used. Five different grids were used to yield a grid independent solution 35x25, 45x30, 50x35, 55x40, and 65x45. A modified SIMPLER scheme for orthogonal curvilinear grid was used. The magnitude of error due to false diffusion for the problem was analyzed. The error was found to peak at the upper boundary of the recirculation area. The results indicated that the main features of the flow were accurately predicted without model adjustments.

Lai and Makomaski (1989) studied a three-dimensional flow pattern upstream of a surface-mounted obstruction. A standard k-ε model, SIMPLE scheme, and a non-uniform grid of 45x24x27 were used. A refined grid of 52x32x27 was tested but found no significant
difference to the coarse grid used. The results showed good agreement with the experimental results.

Han (1989) numerically simulated a three-dimensional turbulent flow around a bluff body in ground proximity. The standard k-ε model was used. A two-step correction procedure known as pressure implicit split operator (PISO) was used in both pressure and velocity corrections. A non-orthogonal curvilinear grid of 61x34x19, 75x43x26, 97x51x31 were tested and 93x51x31 was used. Three different methods, alternating direction implicit method (ADI), Stone's strongly implicit method (STONE), and conjugate gradient (CG) method were tested to solve the algebraic equations. The CG method showed the best performance. The result showed most of the essential features of the flow field around the body. However, the accuracy of drag coefficient prediction requires further development of turbulence modeling. The current k-ε model was found to under predict the base pressure.

Choi and Chen (1990) numerically simulated turbulent flow past finite asymmetric bodies. The standard k-ε model was used with SIMPLER scheme. The calculation domain was 2 times the upper stream and 9.29 times the down stream of the body length of the object simulated. A non-orthogonal grid of 151x26 and 151x36 were used. The results showed good overall agreement between measurements and predictions.

Demuren and Wilson (1994) estimated various errors and uncertainty in computations of two-dimensional separated flows. A laminar flow over a back facing step was used as the test case. The SIMPLE algorithm was used and was the basis for the analysis. The truncation errors of the first order upwind, hybrid, second order upwind, central difference,
and third order upwind schemes were analyzed. The hybrid was the order of 1.9 in truncation error. The discretization error was investigated by Richardson extrapolation method. It was found that Richardson extrapolation method was useful to reduce the discretization error. The error introduced by truncating the outflow domain was investigated. The downstream distance of 7, 10, 15, and 30 times the inlet width were tested. No significance differences were found between the results. The computational grid aspect ratio was investigated. High aspect ratios were found to be more effective in generating accurate solutions in boundary layer region, where high gradient in cross flow direction and low gradient in main flow direction existed. The convergence criterion and its impact on error were also discussed.

Baskaran and Stathopoulous (1992) studied influences of computational parameters on the evaluation of wind effects on the building envelope. The SIMPLE algorithm was used as the based for analysis. The effect of domain size was investigated. The domain sizes of 3x6x3, 6x12x5, 10x20x8, 13x26x10 of the building length (upwind distance x downstream distance x width) were tested. An increase in the domain size was shown to decrease the size of the recirculation zone. Only the first domain size showed a significant difference. The second domain was recommended for use. Grids of 38x20x28, 48x22x32, 58x26x36, 68x30x40, and 78x36x40 were used. The fourth grid was found to be adequate for computation. Mehta (1991) investigated aspects of uncertainty in CFD results. The uncertainties are introduced by a lack of adequate theory, computer model, the unsatisfactory computational accuracy.
Separation flow was also studied by Dennis et al. (1993), Note et al. (1993), and Hung (1991).

3.6. The Numerical Algorithms

From the numerical simulation review, the SIMPLE type of algorithms was the dominate algorithms used. Although this study is not a study about algorithms, a brief review of the algorithms provides a prospective of current advances. SIMPLE (Semi-Implicit-Pressure-Linked-Equations) algorithm was proposed by Patankar and Spalding (Patankar, 1980; Patankar and Spalding, 1972; Spalding, 1972). SIMPLER (Semi-Implicit-Pressure-Linked-Equations-Revised) is a revised version of SIMPLE. SIMPLER converges faster than SIMPLE but requires more memory (Patankar, 1980). Because of the robustness and simplicity, the SIMPLE type of schemes is widely used.

Various modifications were proposed to improve the performance of SIMPLE(R) schemes. Doormeal and Raithby (1984) discussed several modifications to the SIMPLE. The modifications were said to simplify the implementation and to reduce the solution costs. The new scheme was called SIMPLEC. SIMPLEC was reported to perform better than SIMPLER.

The staggered grid used in the original SIMPLE algorithm complicated the implementation of the scheme, especially for the general curvilinear coordinate system. The number of terms in the source term may be overwhelming to implement using staggered grid. Various schemes are proposed for using non-staggered grids. Thiart (1990a) proposed a new
method to avoid the staggered grid used in the SIMPLE algorithm. The source terms were
upwind to avoid the use of a staggered grid. The method was later extended (Thiart, 1990b)
by application of upwind to the terms representing cross stream fluxes in addition to the
upwind of the source terms. The new algorithm was called SIMPLEN. It was found to be in
excellent agreement with the benchmark solution. SIMPLEN was also extended to
cylindrical polar coordinates (Thiart and Backstrom, 1993). But the upwind scheme used for
source terms is a first order accuracy scheme.

Acharya and Monkalled (1989) studied flow problems on non-staggered curvilinear grids.
SIMPLE algorithm was modified as SIMPLEM algorithm. SIMPLEM was found to be
better than SIMPLER in non staggered curvilinear meshes. Kobayashi and Pereira (1991)
extended the SIMPLE algorithm to non-orthogonal, non-staggered grid.

Improving the iteration process to speed up the convergence process is another area of
research effort. Based on SIMPLE, Marek and Straub (1993) developed a fully explicit
iteration scheme MAPLE (Modified-Algorithm-for-Pressure-Linked-Equations). A
combination of under and over relaxation was used to speed up the convergence. The
preliminary results showed better prediction compared with other SIMPLE type of
algorithms. The hybrid of under- and over-relaxation was reported extendable to other
iterative procedures.

Gopinath and Ganesan (1992) demonstrated the use of Orthogonal Array Technique in
numerical simulation to minimize the iterations required. It was found that when doing grid
independence tests, all the under relaxation factors could be optimized on a coarse grid.
One major problem of the iteration scheme (like SIMPLE) is its slow progress to smooth out the low frequency errors, especially when a fine grid is used. To overcome this problem, a multigrid method was proposed. This method uses a series of grids ranging from coarse to fine. The coarse grid allows the removal of low frequency errors quickly. The problem is solved by alternating from a coarse to a fine grid and then from a fine to a coarse grid. Shyy et al. (1993) successfully introduced multigrid computation into SIMPLE scheme for turbulent recirculating flows in complex geometry by using curvilinear coordinates. The CPU time was significantly reduced. Sathyamurthy and Patankar (1994) also introduced a multigrid method for fluid flow problems. A five to 15 times reduction in CPU time was reported.

For many practical problems, the domain of interest may contain regions that are different from each other. The capability of using different zones in the domain can increase the flexibility and efficiency. Shyy and Wright (1994) used multiblock overlapped curvilinear grids in SIMPLE. Issues concerning discontinuous grids, global mass conservation, and block interface treatment are discussed. Majumdar et al. (1992) used a non-orthogonal grid cell centered variable arrangement to improve the flexibility of SIMPLE to use for complex geometry.

As shown in the above review, SIMPLE type of algorithms were studied extensively. Different modifications, especially non-staggered grid, multigrid, and multizone, could significantly simplify the implementation, improve the flexibility, and speed up the convergence process. Many other schemes were also used by researchers to improve the

3.7. Turbulence Modeling

3.7.1. Classification of turbulence models

Much literature on turbulence-modeling is available. White (1991) reported that 100 or more turbulence modeling papers are published every year. In addition, there are three or four annual reviews on the subject. White classified the turbulence model as follows:

Zero equation models: Zero equation models are based on the eddy viscosity concept (Nalllasamy, 1987). The turbulent diffusion terms are related to local gradient of mean flow quantities and mixing lengths. Because of the problem with evaluating the mixing length term, the zero equations model were restricted to relatively simple flows.

One equation model: The one equation model requires the solution of a partial differential equation for the turbulent kinetic energy, \( k \). The turbulent dissipation and eddy viscosity are modeled using mixing lengths. The two well known equation models were proposed by Wolfshtein (1969) and Norris and Reynolds (Reynolds, 1976). The one equation model was reported as no better than the best zero-equation method. It also has the problem of evaluating the mixing lengths as the zero-equation models had. One equation model by itself was not popular (White, 1991). Recently, one equation model has been used as part of the two layer model for near wall region turbulence modeling.
Two equation model: In two equation models, the turbulent energy equation is coupled with a second equation modeling dissipation, turbulent length scale, or some other related quantities. The most widely used turbulence model in engineering calculation is the two-equation $k$-$\varepsilon$ model as shown by Launder and Spalding (1974). Two equation models, especially $k$-$\varepsilon$ models, will be reviewed in detail.

Reynolds stress models: The above turbulence models discussed are based on the assumption of isotropic eddy viscosity. The same values of $v_i$ are taken for different $u'_i, u'_j$ terms. In the Reynolds stress model, each stress term employs a transport equation, which makes Reynolds stress model more universal but computationally expensive. Despite the complexity, the Reynolds stress model does not always give a better prediction than the eddy viscosity method. Pollard and Martinuzzi (1989) did a comparative study of Reynolds stress model and the $k$-$\varepsilon$ model in predicting turbulent pipe flow. It was found that the $k$-$\varepsilon$ model predicted the flow better than the Reynolds stress model. Bradshaw (1989) attribute the problem to the lack of detailed experimental data needed for the modeling. However, the Reynolds stress model was theoretically better than the eddy viscosity model. They are widely used in meteorology modeling (Stull, 1991).

Algebraic stress model: Algebraic model was intended to reduce the complexity of the Reynolds stress model. The differential equations for $u'_i, u'_j$ were replaced by a purely algebraic relation (White, 1991). A comparative study of algebraic model with a $k$-$\varepsilon$ model
by Martinuzzi and Pollard (1989) showed that the algebraic model did not predict the flow as well as the k-ε model.

Large-eddy simulation (LES): In this approach, the large eddy structures are computed through numerical solution of Navier-stokes equations while the small-scale eddy is modeled through turbulence model. The concept is based on the argument that small-scale structures are universal in most flows and can be modeled, while large structures depended on the given flow.

Direct numerical simulation (DNS): DNS tries to resolve all the scales of turbulence by numerical solution of Navier-stokes equations. Since this requires a very fine computational mesh, its use is limited by computer capability to low Reynolds number flows. DNS results have been used to gain in-depth knowledge of the turbulence structure and to calibrate various turbulence models (Reynolds, 1976; Rodi, 1993; Yang and Shih, 1993).

3.7.2. The standard k-ε model and its limitations

Most of the engineering applications involving turbulence modeling uses k-ε two equation models. The high Reynolds number k-ε model as described by Launder and Spalding (1974) is the basis for most of the other versions and is often referred to as the standard k-ε model. The standard k-ε model relies on a wall function for boundary conditions near a solid surface (Launder and Spalding, 1974). The wall function, expressed as Equation 3.4, is basically a dimensionless velocity profile established by analytic and experimental studies for boundary layer flow.
However, the wall function is not universal. It is not valid in many cases of complicated flows including boundary layer flow with adverse pressure gradient and separation flow (White, 1991). The performance of the standard k-ε model for complicated flows is thus poor. Rodi et al. (1986) compared the performances of an one equation model with the standard k-ε model and a low Reynolds number version of the k-ε model under adverse pressure gradient conditions. It was found that the one equation model showed better results than the k-ε models. The result showed that the k-ε models gave a high skin friction coefficient. The generation term in the ε equation had to be increased for the k-ε model to conform with experimental results.

Takemitsu (1990) analytically studied the standard k-ε model for a two dimensional channel flow. The model was shown to have divergent terms. This problem is avoided through the delicate adjustment of model constants. The need for a mathematically well posed k-ε model was pointed out.

3.7.3. Proposed modifications to the standard k-ε model

3.7.3.1. Low Reynolds number models

Wall functions may be avoided by extending the k-ε model to the viscous sublayer adjacent to the wall by adding damping functions. These models are the low Reynolds number models. Jones and Launder (1972) introduced various functions to the constants used in the standard k-ε model. An extra term was also added to the turbulent energy
equation. It was shown that this model can give good prediction for various turbulent flows without the wall functions.

Lam and Bremhorst (1984) proposed a different set of functions to modify the constants in the standard k-\(\varepsilon\) model without modifying the turbulent energy equations. The low-Reynolds number of Lam and Bremhorst was widely used in ventilation air flow predictions (Hoff, 1990; Chen et al. 1990). Reviews of Pollard and Martinuzzi (1989) and Martinuzzi and Pollard (1989) comparing Lam and Bremhorst model with other models also showed results in favor of the Lam and Bremhorst model.

Patel et al. (1985) evaluated the existed low Reynolds number model at that time. It was found that most modifications to the high-Reynolds number k-\(\varepsilon\) models lacked a sound physical basis. From an overall examination of the results, the model of Lam and Bremhorst, Chien (1982), and a few other models were found to be better than others. However, further refinement was suggested if any of the low Reynolds number model is to be used with confidence. Chen and Patel (1988) tried to use a modified Lam and Bremhorst low-Reynolds number model for a separation flow. But the modified model was found to converge too slowly.

Mansour et al. (1987) analyzed low-Reynolds number model of Jones and Launder (1974) and Chien (1982). It was found that the turbulent transport in the \(\varepsilon\) equation is adequate in the region away from the wall but needs modification near the wall. The ad hoc damping functions play an important role in predicting the kinetic energy profile.
More recently proposed low Reynolds number models not only attempt to reproduce the mean properties, but also the turbulence properties of the flow near the wall. Nagano and Hishida (1990) proposed another version of low Reynolds number $k$-$\varepsilon$ model, which improved on the earlier versions (Nagano and Hishida, 1987). The improved model was shown to reproduce strictly the limiting behavior of the wall and free turbulence. It was found that this model was superior to the model of Lam and Bremhorst (1984). It removed many of the defects of the $k$-$\varepsilon$ model pointed out by Patel et al. (1984).

Myong et al. (1988, 1990) proposed a near wall $k$-$\varepsilon$ model considering two characteristic length scales for the dissipation rate, one very near the wall and the other remote from the wall. The models of Launder and Sharma (1974) and Lam and Bremhorst (1984) were used for comparison. The models were evaluated using fully developed turbulent pipe flow and channel flow. It was found that the new model resolved two serious weaknesses common to the previous $k$-$\varepsilon$ models. It correctly predicted the wall-limiting behavior and distributions of eddy viscosity even in regions far from the wall. For the flows studied, the model proposed showed better agreement with the experimental results than the model of Launder and Sharma (1974) and Lam and Bremhorst (1984). An anisotropy low-Reynolds number $k$-$\varepsilon$ model was also proposed and used by Myong and Kobayashi (1991), which was based on the above model and incorporated terms from Reynolds stress models.

So et al. (1991) investigated the low Reynolds number models of Myong and Kasagi (1990) and Speziale (1990). It was found that the existing models incorrectly predicted the
behavior for the dissipation rate for the near wall region. A new set of functions were proposed. The proposed model predicted a similar profile as given by DNS results.

Yang and Shih (1993) proposed another set of functions for the low Reynolds model. In this model, the eddy viscosity was characterized by a turbulent velocity scale and a turbulent time scale. The damping function used in the eddy viscosity was chosen to be a function of $R_y = \frac{k^{1/2} y}{v}$. Hence the model could be used for flows with separation. The proposed model would also be suitable for flows far away from the wall. Turbulent channel flows at different Reynolds numbers and boundary-layer flows with and without pressure gradient were calculated to test the model. Results showed that the model predicted well the experimental and DNS results.

### 3.7.3.2. The approach of Chieng and Launder

From a different approach, Chieng and Launder (1980) introduced detailed near wall modeling in the computation of flow in a pipe expansion. The near wall region between the first grid point and the solid wall was divided into two layers, as shown in Figure 2a. The near wall flow was treated as viscous (both laminar and turbulent viscosity are important) to a distance $y_*$ from the wall and fully turbulent beyond this point. The profiles of velocity, kinetic energy, and shear stress were assumed to be known. Integration can be taken from the wall to the first grid point based on the assumed profiles for the turbulent energy equation. Like the standard k-\(\varepsilon\) model, the first grid point should be at the fully turbulent
a. Two layer treatment

b. Three layer treatment

Figure 2. The two layer and three layer treatments
region. The result showed significantly improved heat transfer coefficient prediction without
the need for fine grid points in the near wall region. The low Reynolds number model of
Launder and Sharma (1974) was also tried with a fine grid in the near wall region but was
found to lead to excessively slow convergence.

Amano (1984) extended the two layer treatment of Chieng and Launder (1980) with two
modifications. The integration based on the assumed profiles was extended to the $\varepsilon$ equation
and a buffer layer was added in between the viscous layer and the fully turbulent layer to
become a three layer treatment. Figure 2b shows the three layers. The computed results
were compared with experimental data and the results using the two layer treatment of
Chieng and Launder (1980). It was found that the three layer treatment did better than the
two layer treatment and the result agreed well with the experimental result in terms of
maximum Nu number prediction.

Djilali et al. (1989) numerical simulated convective heat transfer in two-dimensional
blunt rectangular section using various near-wall treatments. Seven treatments were
considered: the standard $k-\varepsilon$ (STD) model; the two layer treatment (2LK) of Chieng and
Launder (1980); the treatment of Chieng and Launder extended to three layer (3LK); the two
layer treatment extended to $\varepsilon$ equation (2L); the three layer treatment extended to $\varepsilon$ equation
(3L); two-layer model using the standard $k-\varepsilon$ model in the outer layer and one equation
model of Norris and Reynolds (Reynolds, 1976) in the layer near the solid surface (2NR);
two layer model using standard $k-\varepsilon$ model in the outer layer and one equation model of
Wolfshtein (1969) in the layer near the solid surface (2W). The 3L model showed the best
result in terms of maximum Nu prediction. Generally, the four models using Chieng and Launder's approach (2LK, 2L, 3LK, 3L) performed significantly better than the other in terms of maximum Nu prediction. The 2W model predicted maximum Nu better than the 2NR model. The STD model gave the worst performance. The details of flow prediction was not compared. But for the reattachment length, the 2W model gave the best performance.

Thangam and Speziale (1992) evaluated the performances of different turbulence models for flow past a backward-facing step. This study investigated a wide range of models with flow past backward facing step. The treatment of Chieng and Launder (1980) and Amano (1984), the low-Reynolds number model of Spezial (1990), and a new non-linear k-ε model proposed which takes into account the Reynolds stress were considered. A 200x100 non-uniform grid was used and grid refinement study indicated 166x73 mesh yielded result with acceptable error. The predicted reattachment length was used as the main criteria to compare the different models. It was found the nonlinear k-ε model gave a result that was very close to the experimental result. The rest of the models gave similar results which were 12% less than the experimental result. He concluded that the complicated low Reynolds number model did not give significantly better results than the Amano's approach. The larger error reported in literature using standard k-ε model was concluded due to lack of grid resolutions.

In his review of turbulent models, Nallasamy (1987) concluded that the multi-layer treatments are useful for improving the prediction of wall heat transfer rates. However, it did not significantly improve the flow field predictions.
3.7.3.3. Replacing ε equation

One of the perceived causes of problems with k-ε model is the ε equation. It has no exact (or analytical) solution at the solid boundary (Wilcox, 1989). From this perspective, different equations were proposed to replace the ε equation. Wilcox (1989) provided a comprehensive and critical review of k-ε two equation models. He found that conventional k-ε models are inadequate for boundary layers in adverse pressure gradient conditions. Using wall functions masks the shortcomings of such models. He proposed to use ω to replace ε. ω is defined as,

$$\omega = \frac{\varepsilon}{\beta k}$$ 3.7

where,

$$\beta = \text{constant}$$

$$k = \text{turbulent kinetic energy}$$

The equation for ω was proposed to be similar with ε equation. The ω has an analytical solution at solid surface. The attached boundary layer flow with adverse pressure gradient, compressible boundary layers, and free shear flows were simulated using this model and accurate results were reported. To show the performance of the k-ω model, Wilcox (1993) compared the k-ω model with six low Reynolds number k-ε models for boundary layer flow of different pressure gradients. He found that k-ε models with $y^+ < 0.1$ for the first grid near the wall slowed the convergence. For the k-ω model, the analytical solution of ω was used for all points with $y^+ < 2.5$. The average value of $y^+$ closest to the surface was 0.2. Results
showed that the $k-\omega$ model was better than the $k-\varepsilon$ models, especially for flows with adverse pressure gradient. The model was also shown to be able to predict the transition between laminar and turbulent flows (Wilcox, 1994).

Mentor (1992) studied the influence of free stream values on $k-\omega$ turbulence model. He found that the $k-\omega$ model is quite sensitive to free stream values. That conclusion was confirmed by Wilcox (1993). The $k-\omega$ model requires a well-known free stream condition. Mentor (1992) compared the performance of some of the turbulence models for attached and separated adverse pressure gradient flows. The $k-\omega$ model was one of the models used and the result from the $k-\omega$ model was in good agreement with the experimental. However, the best performance was from Johnson-King model (Johnson and King, 1985). To avoid the sensitivity of the $k-\omega$ model to free stream condition, Mentor (1994) proposed to use the $k-\omega$ model in the inner region of the boundary layer and switches to the standard $k-\varepsilon$ model in the outer region and free shear flows. Flat plate boundary layer flow, adverse pressure gradient flow, backward-facing step flow, and a transonic bump flow were tested. The results showed the newly proposed model performed well and improved on the original model.

The $k-\omega$ model of Wilcox (1989) was also used by Liu and Zheng (1994) for flat plate flow and cascade flow. The result showed good agreement with experimental data. The transition from laminar to turbulent in the flat plate flow was also reported in good agreement with the experimental results.

Speziale et al. (1990) studied some of the low Reynolds number models and the $k-\omega$ model proposed by Wilcox (1988). It was found that the $k-\omega$ model is missing an exact
viscous cross diffusion term. A new k-τ was proposed by making use of the ideas of the k-ω model and low Reynolds number model of Myong and Kasagi (1990). The results showed the new k-τ model performed better for the flow studied. However, it was concluded that further tests and possible refinements were required. So et al. (1991) compared the k-τ model, low Reynolds number by Myong and Kasagi (1990), and a newly proposed model. It was found the k-τ model behaved similarly with the model by Myong and Kasagi.

Goldberg (1994) also proposed a two equation model using $R = k^{2/3} \varepsilon$ to replace $\varepsilon$.

### 3.7.3.4. Two layer approach

In the two layer approach, two types of models are used, one in the near wall region and one in the outer and free stream region. Usually for the near wall region, a one equation model is used and for the outer region, a two equation model is used.

Chen and Patel (1988) studied the separation flow. A two layer model using the one equation model of Wolfshtein (1969) for the near wall region and the standard k-ε model for the outer region was used. The near wall region was the region of,

$$R_y < 250$$  \hspace{1cm} \text{3.7}$$

where,

$$R_y = k^{1/2} y / \nu$$

$k$ = the turbulent kinetic energy

$y$ = the distance to the solid surface
The rest of the computation domain was treated as the outer region. The standard k-ε model and a modified version of low Reynolds number model of Lam and Bremhorst (1984) were also used. The result indicated that a two-layer approach was quite successful in economically resolving the most important features of complex flows. It was found that from computational perspective, the two-layer model was easier to implement, with relatively few grid points in the wall layer compared with the low Reynolds number model. The two layer approach was also found quite insensitive to grid spacing. The low Reynolds number model was found to not have as good a convergence behavior as the two layer model. It was concluded that the two layer model out performed the low Reynolds number model studied from a numerical and a physical view point.

Patel et al. (1991) used the two layer model by Chen and Patel (1988) to solve the flow in a channel with a wavy wall. The first grid was located at $y^+$ around 0.01 with 99 grid points in the cross flow direction. The results showed that the two layer model was quite successful in calculating flows with multiple separations and reattachments with a reasonable amount of CPU time. The model was also able to capture most of the important physical features of such a flow. The results also demonstrated the breakdown of the wall function before the onset of separation for this type of flows. It was suggested not to use wall function for such flows.

Djilali et al. (1989) used two layer models and other models to study the heat transfer of a blunt rectangular plate. For the near wall region, both the one equation model of Norris and Reynolds (Reynolds, 1976) and Wolfshtein (1969) were used. It was found that the two layer
models were slightly better than standard k-ε model in predicting the maximum Nu number and the reattachment length. But it was not as good as the model of Amano (1984) and Chieng and Launder (1980) in predicting the maximum Nu number. The one equation model by Wolfshtein performed slightly better than that by Norris and Reynolds.

Rodi et al. (1993) proposed a two layer model based on DNS and experimental data. The traditional one equation model (referred to as TLK model) of Norris and Reynolds (Reynolds, 1976) using the square root of turbulent kinetic energy as velocity scale was studied and a new one equation model based on $(\overline{v'^2})^{1/2}$ as velocity scale (referred to as TLV model) was proposed. Using $(\overline{v'^2})^{1/2}$ as the velocity scale was first proposed by Durbin (1991), who showed that $(\overline{v'^2})^{1/2}$ provided a better profile of turbulence dissipation in the near wall region. The DNS and experimental results in the near wall region of channel flows and boundary flows were studied to find the correct near wall behavior. Empirical relations were derived from DNS and experiment data for $(\overline{v'^2})^{1/2}$ and the length scales. The proposed relations was valid for $y^+$ up to 50 and was reported as $\nu/\nu$ of about 16. A developed channel flow, a boundary layer flow over a flat plate, and a flow over backward-facing step were used as the test cases for the model. For the backward-facing step flow, TLK and TLV model were found to give almost the same result but TLV was reported to have a numerical advantage over TLK model by requiring less iterations to converge. The reported reattachment length was 5.7 compared to a 6.2 experimental result and 4.8 using the standard
k-ε model. For the other two types of flows, the TLV model was reported to show better agreement with experimental and DNS result than the TLK model.

Mentor (1994) proposed to use the k-ω model of Wilcox (1989) in the inner region of the boundary layer and to use the standard k-ε model in the outer region and free shear flows. Flat plate boundary layer flow, adverse pressure gradient flow, backward-facing step flow, transonic bump flow and other flows were studied. The results showed the newly proposed model performed well and improved on the original model.

The multilayer approach was also used by Goldberg (1992a, 1992b).

**3.7.3.5. Incorporating Reynolds stress terms**

Two equation k-ε models and other eddy viscosity models are also incorporating the Reynolds stress terms in the equations to avoid the deficiencies of the traditional eddy viscosity models. Johnson and King (1984) incorporated the velocity scale of maximum Reynolds shear stress into an ordinary differential equation of kinetic energy. The model was shown to give a good prediction for flows under adverse pressure and separation flows. Johnson (1986) improved on the model of Johnson and King (1984). A series of flows of transonic, inviscid-viscous interactions with varying degrees of separation were tested. The calculated results were compared with experimental results and were shown to be in excellent agreement. Johnson and Colacley (1990) also showed similar results using the model. Mentor (1992) used various models, including the Johnson-King model and k-ω model of
Wilcox (1988) to study the boundary layer flows including flows with separation. The Johnson-King model was found to be the best model for the flow studied.

Myong and Kasagi (1990) proposed an anisotropy low Reynolds number k-ε model by incorporating terms from the Reynolds stress model. Myong and Kobayashi (1991) showed that the model could be used to predict the flow in a square duct. Thangam and Speziale (1991) also found that when the standard k-ε model was modified to include an anisotropy eddy viscosity, the prediction of flow past a backward-facing step improved significantly. Thangam (1992) proposed to improve the k-ε model using the algebraic stress model. The Cμ term in the k-ε model was modified based on an algebraic stress model. The proposed model was used to predict the flow past a backward-facing step. The model was shown to accurately predict the dominant features of the flow, namely the size of the separation bubble, the mean velocity, and the turbulence stress.

Some of the other studies of turbulence modeling include Caille and Schetz (1993), Walker et al. (1988), Thomas and Hasani (1992), So et al. (1990), Chapman and Kuhn (1986), and Yankhot (1992).
4.1. Problem Description

The tank studied is shown in Figure 1. An open round manure storage tank of diameter $D$ and height $H$ was sitting on the ground. The wind speed was $u_0$ at the top of the computation domain. The manure depth was $h$. The odor concentration at the manure surface (the odor concentration in the air that was infinitely close to the manure surface) was $c_0$, which took into account the bio-processes in manure. The objective was to find out the odor emission rate under that condition. For this study, the following parameters were used: $D=16$ m; $H=8$ m; $u_0=5$ m/s; $c_0=5,000$ ou. The Reynolds number using the tank height as the length scale is $2.7 \times 10^6$.

4.2. Selection of Method

Four methods were considered for possible use to study the odor emission rate from the tank. They were: an open bottom wind tunnel floating on manure, a box covering the tank, similitude study using wind tunnel, and numerical simulation.

4.2.1. An open bottom wind tunnel floating on manure

In this approach, an open bottom wind tunnel is used to float on the manure surface. A fan is used to provide air movement in the tunnel and provide air exchange with outside air. An active charcoal filter is placed in the inlet to remove any odor from the air entering the
tunnel. The exhaust air flow rate and the odor concentration in the exhaust can be measured. The odor emission rate can then be calculated. This approach was discussed and used by Homans (1988). It was used to study the odor and ammonia emission rate from manure land spreading. Bode (1991) also mentioned using an open bottom wind tunnel to measure the odor emission rate from a full size manure storage tank. No details or results were given.

This approach is simple and provides a rough estimate of the emission rate from the tank, however, the flow in the wind tunnel has little resemblance to the actual flow pattern in the tank. It is also difficult to relate the air speed in the tunnel to the wind speed. It is difficult for this method to meet the requirement of dispersion modeling.

4.2.2. A box covering the tank

This method uses a box to cover the tank. A fan is used to create air movement in the tank. It also provides air exchange with the outside. The exhaust air flow rate and the odor concentration in the exhaust can be measured. The odor emission rate can be calculated. This approach was used by Bode (1991) for an odor and ammonia emission rate study.

Compared with the first method, the air flow in the tank by this approach may be more like the air flow in an actual open tank, thus a better estimation of the emission rate may be obtained. The flow pattern will still be different from the flow in the open tank and the problem of relating the air flow speed with the ambient wind speed is not solved.
4.2.3. Similitude study

This approach uses a scaled model of the tank in a wind tunnel, if possible an atmospheric surface layer wind tunnel, to simulate the air flow in the tank. If the similitude criteria are kept the same for the prototype and the model, a similitude exists between the model and prototype. The data measured in the model can be extended to the full size prototype. Similitude is widely used for atmosphere pollution and building pressure distribution.

The use of similitude was found not to be feasible. The reasons were:

1) Difficulty in measuring the emission rate in the scaled model: Three ways were considered to simulate the manure surface in the scaled tank: using manure in the scaled tank; using other chemicals to replace manure in the tank; and using a solid surface with evenly distributed holes to simulate the manure surface.

*Using manure in the scaled tank.* Measuring the emission rate from the scaled model can only be done by collecting all the exhaust air and measuring the concentration in the exhaust or by ensuring the exhaust air is well mixed and measuring the concentration in the sample. Both are not practical and the mixed exhaust air may have concentration too low to detect.

*Using other chemical in the tank.* If significant amount of evaporation takes place, the estimate of the emission rate is possible by weighing the liquid before and after the experiment. This requires a highly evaporative liquid. However, these kinds of chemicals also has a high diffusion coefficient, which would significantly reduce the impact of
convection and distort the transport process. As shown later by the numerical simulation, changes in diffusion coefficient can significantly change the emission rate.

**Solid surface with evenly distributed holes.** The surface may or may not be covered with porous media. However, with and without porous media, the viscous sublayer near the surface would be destroyed. Without the viscous sublayer, the emission rate would be many times higher than with the viscous sublayer. The same destruction also occurs if the air is bubbled through a liquid, like water.

2) **Possible error due to relaxing some similitude criteria.** The dimensional analysis showed that the Reynolds number and Peclet number are among the similitude parameters needed to meet if a similitude can be reached (Szvcs, 1980; Snyder, 1972). This requirement must be relaxed because of the high air velocity requirement if it is not. The relaxing is used in similitude studies of the ABL boundary layer involving pollution and building pressures. Whether it can be ignored in this problem is not known because the source is located in the viscous sublayer region. The result of numerical simulations in this study showed that Re and Pe changes in a wide range of Re (Pe) number values resulted in relatively small change in emission rate. But a big difference (difference in magnitude) still results in significant differences in emission rates.

Although water was used in some applications of a similitude to replace air, the use of water or other media for this similitude study was quickly eliminated. If water would be used, another chemical with the right density and diffusion coefficient must be found. There
would also be problems in measuring the concentration of the chemical and simulating the manure surface.

3) **Uncertainties using similitude study.** Although similitude is accurate on a theoretical basis, in practice it can be difficult to accomplish. The similitude of room air motion under isothermal condition, which is a relatively simple flow with no mass transfer and involving relatively small scale factors, was proven to be difficult to get the adequate accuracy (Zhang, 1991). How to ensure similitude is reached and the result can be scaled up remains to be a question.

The above reasons, especially the first one, made the similitude study not feasible.

4.2.4. **Numerical simulation**

Numerical simulation solves the governing equations for the flow and conservation of the species equation numerically. Numerical simulation of the transport process in the air is currently feasible with reasonable accuracy. Although the flow involved in this study includes separation, the literature review showed that separation flow can be simulated with good accuracy (Boris, 1989; Paterson and Apelt, 1989).

As pointed out by Kot (1989) and Douglass and Ramshaw (1994), numerical simulation is not developed such that it can be used as a black box. The computational domain, the computational grid, and the turbulence model used may significantly impact the result and in extreme cases may result in useless results. Because of the current computer capability and the current understanding or lack of understanding of the problem, significant simplifications
and assumptions may also be made to solve the problem. Because of these uncertainties, numerical simulation has to be used with experimental verification except limited simple cases to make sure the numerical simulation is used properly.

4.2.5. Method selected

Based on the above analyses, the numerical simulation approach with experimental verification was selected. The first two methods did not have the potential to meet the demands of dispersion modeling and the similitude study was not feasible.

The air flow and odor concentration inside and in the vicinity of the tank was numerically simulated based on the wind speed, tank dimensions, and manure depth. The odor emission rate was calculated based on the numerical simulation results. The experiment was then carried out to verify the numerical prediction. However, direct experimental verification of the full scale tank was not feasible because of the size of the tank. Instead, a wind tunnel was constructed to test the flow in the scaled tank. Separate numerical simulations were carried out based on the wind tunnel configuration to compare the experimental flow pattern with the predicted flow pattern. If the prediction is comparable with the experimental results, it is assumed that the numerical simulation can be used to model the flow in the actual tank. This approach, if proven successful, can also be used to predict the emission rates in the lagoon.
4.3. The Assumptions

Odor emission from an open tank can be very complicated. To numerically estimate the emission rate, the following assumptions were made to simplify the problem:

1. The air flow and odor emission process were at steady state.
2. The manure surface and the air above it were at the same constant temperature.
3. The odor concentration at the manure surface was the same for the entire surface.
4. All the odor ingredients as a group behaved as one gas.
5. No other source or sink existed for odor except the manure surface as a source. No chemical reaction took place in the domain of interest.
6. The manure surface was stationary.
7. The air can be treated as incompressible.

4.4. Simplification from a Three-Dimensional to a Two-Dimensional Problem

The flow in an open round tank is without question three dimensional. However, a three-dimensional numerical simulation is currently not feasible because of the available computer capacity and large number of grid points needed to solve the problem. For the two-dimensional version of the problem, four megabytes of memory and about four hours of CPU time were required to solve the problem. The computer output was approximately two megabytes. Considering another direction would have required many times (40 or more) that amount and was clearly not feasible.
To simplify the problem, the circular tank was divided into four sections of rectangular areas, as shown in Figure 3. In Figure 3, the numbers in parenthesis is the percentage of area the rectangle has in relation to the area of the half tank. Each rectangle was then treated as a section of infinitely long rectangle of width, W, and was treated as two dimensional. Instead of solving one three dimensional problem, four two dimensional problems were solved. Figure 4 shows the two dimensional problem that was actually solved, which has two walls of the same height H, and placed at a distance of W from each other. W values were 0.6H, 1.2H, 1.6H, and 2H, respectively, for the four rectangles. H is the tank height.

4.5. Calculation Domain

To correctly predict the flow, the computation domain has to be selected adequately. The computation domain was chosen based on the results of Baskaran (1992), Thangam and Spezial (1992), and Djilali (1990). Baskaran studied the influence of calculation domain on the building envelope. The domain sizes of 3x6x3, 6x12x5, 10x20x8, 13x26x10 of the building length (upwind distance x downstream distance x width) were tested. Except the first domain size, the others did not show a significant difference. Thangam and Special (1992), who studied the flow past a backward-facing step, reported that 30 step heights downstream was needed to ensure that the local error is in the same order as the interior values. Djilali (1990) studied the flow around a blunt plate. He reported that changing the computation domain from 15 to 9 times the plate thickness upstream of the plate as the upstream boundary had no noticeable effect on the flow in the recirculation region. A 0.3%
increase in the bubble length was reported when 7.5 times the plate thickness was used. The
outflow boundary greater than 11 times the plate thickness from the separation point showed
no impact on the bubble. 1.0% larger in bubble was reported when the location was 8 times
the plate thickness.

In this study, the calculation domain was chosen from 6H upstream of the tank center to
21H down stream of the tank center in horizontal direction and from ground to 11H high.
Results using a larger computation domain of 8x25x15 times the tank height (upstream x
down stream x vertical) for this study did not shown any noticeable effect on the results.

Figure 3. Dividing the tank into rectangles
Figure 4. The computation domain after simplification
CHAPTER 5. MATHEMATICAL MODEL

The flow in the simplified tank is a steady state, incompressible, isothermal turbulent flow. The governing partial differential equations and the turbulence model will be discussed in this section. A more detailed discussion on the governing partial differential equations can be found in White (1991) and Hinze (1975).

5.1. The Time Averaged Governing Equations

5.1.1. The equations for laminar flow

The governing equations for two dimensional, laminar, steady state, isothermal flow are (White, 1991):

Continuity equation:

$$\frac{\partial (\rho u_i)}{\partial x_i} = 0$$

Momentum equations:

$$\frac{\partial (\rho u_i u_j)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{\partial p}{\partial x_j} \right)$$

Conservation of species:

$$\frac{\partial (\rho u_j c)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( D_c \frac{\partial c}{\partial x_j} \right) + S_c$$

In the Equations, i=1, 2 and j=1, 2.
5.1.2. The time averaged governing equations

Using the Reynolds decomposition for turbulent flow, the flow variables are replaced by time averaged values plus fluctuations on the average. Assuming constant density, the variables are expressed as:

\[
\begin{align*}
\bar{u} &= \bar{u} + u' \\
\bar{v} &= \bar{v} + v' \\
\bar{p} &= \bar{p} + p' \\
\bar{c} &= \bar{c} + c' \\
\bar{\rho} &= \bar{\rho}
\end{align*}
\]

5.4

The time averaged equations were derived by inserting Equation 5.4 into the continuity, momentum, and conservation of species equations. The equations are then averaged by observing the following rules of averaging:

\[
\begin{align*}
\bar{f}' &= 0 \\
\bar{f} &= \bar{f} \\
\bar{f}'g &= 0 \\
\frac{\partial f}{\partial g} &= \frac{\partial \bar{f}}{\partial g} \\
\bar{fg} &= \bar{f}g + \bar{f}'g'
\end{align*}
\]

5.5

The time averaged governing equations are:

Continuity equation:

\[
\frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_i} = 0
\]

5.6
Momentum equations:

\[ \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \left( \frac{\partial \bar{u}_j}{\partial x_j} + \frac{\partial \bar{u}_i}{\partial x_i} \right) - \rho u_i' u_j' \right) - \frac{\partial \bar{p}}{\partial x_j} \]  

Conservation of species:

\[ \frac{\partial (\rho \bar{u}_j \bar{c})}{\partial x_j} = \frac{\partial}{\partial x_j} \left( D_c \left( \frac{\partial \bar{c}}{\partial x_j} - \rho u_j' c' \right) \right) + S_c \]

where,

\[ D_c = v/Sc \]

\[ Sc = \text{Schmidt number} \]

In the Equation 5.6, 5.7, and 5.8, i = 1, 2 and j = 1, 2.

The continuity, momentum, and conservation of species equations are not closed equations because of the turbulent fluxes terms in the momentum, \(-\rho u_i' u_j'\), and conservation of species equation, \(-\rho u_j' c'\), are unknown. Turbulence modeling is needed to calculate the turbulent fluxes and close the equations. There are generally two ways to model turbulence: eddy viscosity and Reynolds stress approach. The eddy viscosity was less complicated and computationally less expensive and generally comparable to Reynolds stress model in prediction accuracy. Thus the eddy viscosity approach was selected.

The eddy viscosity approach follows the suggestion of Boussinesq that the turbulent stress and fluxes can be express using mean quantities and eddy viscosity,

\[ -\rho u_i' u_j' = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} k \rho \delta_{i,j} \]
\[-\rho \overline{u_i u_j} c = \rho D_i \frac{\partial \bar{c}}{\partial x_j}\]  \hspace{1cm} 5.10

where,

\[\delta_{i,j} = \begin{cases} 
1 & \text{where } i = j \\
0 & \text{where } i \neq j
\end{cases}\]

\[D_i = \nu_t/\sigma_c\]

\(\mu_t\) is the eddy viscosity, and \(D_i\) is the turbulent diffusion coefficient. With this treatment, \(\nu_t\) is the only thing left that needs turbulence model calculate.

5.2. Selection of Turbulence Model

5.2.1 Model selection

The standard \(k-\varepsilon\) two equations model (Launder and Spalding, 1974) was the most popular one used in engineering calculation for \(\mu_t\) calculation. The standard \(k-\varepsilon\) model uses the wall functions as the boundary conditions for solid surface. However, the wall functions are not valid for complex flow including separation (White, 1991; Patel et al, 1991; Nallasamy, 1987). Many models were proposed and many of them showed significant improvement over a standard \(k-\varepsilon\) model. Some of the recently proposed models that showed significant improvement included the low Reynolds number model of Myong and Kasagi (1990), the three layer treatment of Amano (1984), \(k-\omega\) model of Wilcox (1988) and Mentor (1994), two layer model of Chen and Patel (1988) and Rodi (1993), and the model proposed by Thangam (1992).
For this study, the two layer model of Rodi (1993) was selected. It uses the standard k-ε model for outer layer and free stream and a one equation model for the layer close to the wall. It is well known that the one equation model is generally comparable in accuracy with the two equation models and are more reliable than the two equation models for complex flow situations (including separation). The standard k-ε model has proven its suitability for free shear flows. Combining these two can use the advantages of both models and avoid some of the weaknesses. The implementation of the two layer model is also relatively simple.

5.2.2. Rodi's two layer turbulence model

Since the two layer model uses the standard k-ε model for the outer layer and the free stream and a one equation model for the layer close to the solid wall, the standard k-ε model and the one equation model used are discussed in detail in this section.

5.2.2.1. The standard k-ε model

The standard k-ε model was shown by Launder and Spalding (1974). In this model, the turbulent viscosity (eddy viscosity) μₜ is calculated as a function of turbulent kinetic energy, k, and turbulence energy dissipation, ε,

\[ \mu_t = C_\mu \rho k^2 / \varepsilon \]  \hspace{1cm} 5.11

where,

\[ k = \frac{1}{2} \left( \overline{u_i^' u_j^'} \right) \]  \hspace{1cm} 5.12
\[ \varepsilon = \mu \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_j}{\partial x_i} \]

5.13

\[ i, j = 1, 2 \]

The problem of determining the turbulence stress is to determine \( k \) and \( \varepsilon \). Two additional equations, one for \( k \) and one for \( \varepsilon \), were needed in the standard two equation model. The exact transport equation for \( k \) and \( \varepsilon \) can be obtained by manipulating the momentum equations. The two equations are (Launder and Spalding, 1974):

\[
\frac{\partial (\rho k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\mu}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \rho \varepsilon \quad 5.14
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\mu}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + c_1 \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - c_2 \rho \frac{\varepsilon^2}{k} \quad 5.15
\]

The values of the constants were shown in Table 1. More detailed discussions on the two equations can be found in Horlow and Nakayama (1967) and Markatos (1986).

The standard \( k-\varepsilon \) model assumes the flow is fully turbulent. For flows involving solid boundary, it is not capable of dealing with the region close to the solid surface. To circumvent the problem, the first computational point near the wall is to be located outside

<table>
<thead>
<tr>
<th>( c_\mu )</th>
<th>( c_1 )</th>
<th>( c_2 )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>
the viscous sublayer region in the standard k-ε model. A function that has been established by analytical and experimental studies were used to bridge the region from the solid wall to the first grid point. This relation is the wall function.

The wall function can be derived by one dimensional flow assumption in the near wall region. For the region near a wall, the changes in variables along the wall is small. Thus, for the near wall region, the flow can be assumed as Couette flow, i.e. one dimensional flow. It can be shown that for this region, the velocity can be expressed as (Hoff, 1990; Chen, 1988)

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

\[ \text{Equation 5.16} \]

where,

\[ u^* = \bar{u} / \nu^* \]
\[ y^* = y \nu^* / \nu \]
\[ \nu^* = \left( \tau_w / \rho \right)^{1/2} \]
\[ \kappa = 0.41 \]
\[ B = 5.0 \]

\[ \tau_w \] is the wall friction. It can be calculated as \( \tau_w = \frac{1}{2} \rho \kappa \) for standard k-ε model (Launder and Spalding, 1974). Equation 5.16 is also referred to as log-law or law of wall. It can also be derived by dimensional analysis (White, 1991). Experimental studies showed that Equation 5.16 is a great success for attached boundary layer flows. The wall function was shown to be valid for the range of \( y^* \) from about 35 to 350 for attached flow on a flat plate (White, 1991). However, for standard k-ε model, the log-law is treated as valid for \( y^* \geq 11.3 \) (Launder and Spalding, 1974).
The wall function is part of the boundary conditions for boundary condition near the solid surface. The other boundary conditions at the solid surface are (Hoff, 1990),

\[ v = 0 \]
\[ \frac{\partial k}{\partial y} = 0 \]
\[ \varepsilon \to \infty \]

In practice, the \( \varepsilon \) for the grid points adjacent to the solid surface is calculated

\[ \varepsilon = \frac{c^{3/4} k^{3/2}}{\kappa y} \]

5.2.2.2. The one equation near-wall turbulence model of Rodi

As shown in the previous discussion, the standard k-\( \varepsilon \) model relies on the wall functions for the near wall region. However, the wall function is not valid for separation flows. For complex flows including separation, the one equation model, which does not need the wall function, was found to be more reliable or comparable than the standard k-\( \varepsilon \) model (White, 1991; Rodi and Scheuerer, 1986). Detailed discussions on one equation models were available in Reynolds (1976), Wolfshtein (1969), and Rodi et al. (1993).

The basic approach of one equation model is to use the transport equation of the turbulent kinetic energy equation, Equation 5.14, for kinetic energy calculations. The turbulent energy dissipation calculation is based on an algebraic formula using a length scale. The length scales are based on the distances to the solid surface with damping functions.
The one equation model was proposed to form a two layer model with the standard \( k-\varepsilon \) model to take advantages of both models and avoid the draw backs. This approach was used by Chen and Patel (1988), Patel et al. (1991), Djilali et al. (1989), Goldberg (1991), Rodi et al. (1993), and Mentor (1994) with success. The model proposed by Rodi et al. (1993) was used for this study.

The main difference between the one equation model proposed by Rodi (1993) and other one equation models is that this model uses \( (v'^2)^{1/2} \) as velocity scale for characterizing the turbulent motion in the place of \( k^{1/2} \) as used in the other one equation models. Three advantages were shown by Rodi using this new scale: a better \( \varepsilon \) profile for near wall region, less damping needed, and better convergence behavior. The following equations were proposed for the calculation of turbulent viscosity, \( \mu_t \); and turbulence dissipation, \( \varepsilon \),

\[
\mu_t = \rho \left( \frac{\overline{v'^2}}{l_{u}} \right)^{1/2} l_{u} \tag{5.19}
\]

\[
\varepsilon = \frac{\left( \frac{\overline{v'^2}}{l_{u}} \right)^{1/2} k}{l_{\varepsilon}} \tag{5.20}
\]

where,

\[
l_{u} = c_{l,u} y
\]

\[
l_{\varepsilon} = \frac{1.3y}{1 + \frac{2.12\nu}{\left( \frac{\overline{v'^2}}{l_{u}} \right)^{1/2} y}}
\]

\[
c_{l,u} = 0.33
\]

The calculation of \( \left( \frac{\overline{v'^2}}{l_{u}} \right)^{1/2} \) is based on curve fitting through the DNS results. The fitted curve is,
\[
\frac{\langle v'^2 \rangle}{k} = 4.65 \times 10^{-5} y'^2 + 4.00 \times 10^{-4} y^* \quad 5.21
\]

Where,

\[
y^* = \frac{k^{1/2} y}{v}
\]

Equation 5.21 was shown to describe the DNS data well for \( y^* \) up to 60.

Equation 5.19, 5.20, and 5.21 are all empirical equations based on the experimental data and DNS data. For the one equation model, no special boundary conditions near the solid surface is needed.

5.2.2.3. The two layer model

In the two layer model, the one equation model described above was used as a component of a two-layer model in the near-wall region while the flow outside this region is calculated with the standard k-\( \varepsilon \) model. The standard constants as listed in Table 1. To predicted the channel flow better, Rodi et al. (1993) used \( \sigma_k = 1.3 \) instead of 1.0, which is the standard value. The transfer from one equation model to the two equation model was recommended at \( y^+ \) of about 50, which is at \( \nu_\lambda / \nu \) of about 16. In this study, \( y^+ = 50 \) was set as the transition point.

Combining these two models can use the advantages of both models and avoid some of the weaknesses. The implementation of the two layer model is also relatively simple.
5.3. The Diffusion Coefficients

In the species conservation equation, Equation 5.8, the turbulent flux term, \(-\rho u'c'\), was calculated according to Equation 5.10 following the suggestion of Boussinesq. Two parameters, Sc and \(\sigma_c\), need to be decided. The Schmidt number, Sc, decides the laminar diffusion coefficient and is the property of the species and the media, in this case, odor and air, respectively. Odor from the manure storage facility has many ingredients and many of them are not even identified (Ritter, 1989). The exact Schmidt number for odor as one gas may never be known. A Sc number of 1.0 was used for this study. The selection of Sc was based on the fact that the inorganic components NH3 and H2S have Sc values smaller than one and the other organic ingredients were heavier and may have larger Sc (Reid, 1977). The \(\sigma_c\) value is mainly dependent on the flow and less dependent of the species. For most of the gases with Sc number close to 1, the \(\sigma_c\) is 1 (Treybal, 1980). \(\sigma_c=1.0\) was used.

5.4. The Generalized Partial Differential Equations

The time averaged equations of continuity, momentum, and conservation of species and the equations from turbulence modeling can be put in one general form. Only one discretization is needed for numerical simulation with the generalized form. For simplicity, the over bar notation for the time averaged mean quantities is dropped. The general form of the partial differential equations for the present flow configurations can be written as,

\[
div(\rho\bar{v}\phi) = div(\Gamma_{eff} \nabla \phi) + s_i
\]

5.22
Table 2. Summary of equations

<table>
<thead>
<tr>
<th>Equation</th>
<th>$\phi$</th>
<th>$\Gamma_{eff}$</th>
<th>$S\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Momentum</td>
<td>$u_i$</td>
<td>$\mu + \mu_t$</td>
<td>$-\partial p/\partial x + \partial/\partial x_j (\Gamma_{eff} \partial u_j/\partial x_i)$</td>
</tr>
<tr>
<td>Turbulent energy</td>
<td>$k$</td>
<td>$\mu + \mu_t/\sigma_k$</td>
<td>$\rho (Gt-e)$</td>
</tr>
<tr>
<td>Dissipation†</td>
<td>$\varepsilon$</td>
<td>$\mu + \mu_t/\sigma_e$</td>
<td>$\rho (c_1 \varepsilon * Gt/k - c_2 * \varepsilon^2/k)$</td>
</tr>
<tr>
<td>Concentration</td>
<td>$c$</td>
<td>$\mu/Sc + \mu_t/\sigma_c$</td>
<td>0</td>
</tr>
</tbody>
</table>

† For area that is not close to the solid surface only.
‡ $G = u_t [(\partial u_t/\partial y + \partial v/\partial x)^2 + 2(\partial u_t/\partial y)^2 + 2(\partial v/\partial y)^2]

Table 3. The constants used in the equations

<table>
<thead>
<tr>
<th>$c_\mu$</th>
<th>$c_1$</th>
<th>$c_2$</th>
<th>$\sigma_k$</th>
<th>$\sigma_e$</th>
<th>$Sc$</th>
<th>$\sigma_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

This generalized equation is in the form used in the SIMPLE scheme, which was used in this study. The individual equations are summarized in Table 2 and the constants used in the equations are listed in Table 3.

5.5. Non-Dimensionalization of the Equations

The following non-dimensional variables were defined to non-dimensionalize the governing equations:
\[ u^* = u/u_0 \]
\[ v^* = v/u_0 \]
\[ x^* = x/H \]
\[ y^* = y/H \]
\[ p^* = p/(\rho u_0^2) \]
\[ c^* = c/c_0 \]
\[ v^* = v/(\rho u_0 H) \]
\[ D_c^* = D_c/(\rho u_0 H) \]
\[ \varepsilon^* = \varepsilon/(u_0^2/H) \]
\[ l\mu^* = l\mu/H \]
\[ v^* = v/u_0 \]
\[ l_c^* = l_c/H \]

\( H \) is the tank height. \( u_0 \) is the wind speed, \( \rho \) is the free stream air density, \( c_0 \) is the odor concentration at the manure surface. Using the defined variables, the non-dimensionalized equations are the same as the original equations except that every variable is non-dimensionalized. Dropping the * notation, the equations are exactly the same and thus are not repeated here.
5.6. Boundary Conditions

Table 4 is a list of all the boundary conditions. At the upstream boundary AB (see Figure 4), the condition was assumed to be the undisturbed wind flow with a wind velocity profile of 1/7 power law (Equation 3.1) and turbulent intensity of 15% (Chok, 1988). The turbulent dissipation was evaluated based on the expression from Launder and Spalding (1974), which is,

$$
\varepsilon = \frac{c_u k^{3/2}}{l}
$$

where $l$ is a length scale. This treatment was used by Murakami and Mochida (1989), Majumdar and Rodi (1989). The turbulent kinetic energy and dissipation were assumed to be constant across this boundary and a length scale of 0.9 was used. At the down stream boundary CD, the velocity profile was assumed to have recovered to the original wind velocity profile and the other quantities were assumed to have zero gradient in the $x$ direction. At the top boundary BC, the velocity was assumed to be unaffected by the tank and the other variables were assumed to have zero gradient in the $y$ direction. At the solid surfaces, including the ground and the tank walls, no-slip boundary conditions were imposed for velocity and no absorption was used for odor concentration. The same conditions were used for manure surface except the odor concentration was assumed to be known.

Please note that in Table 4, the values were non-dimensionalized.
Table 4. The boundary conditions

<table>
<thead>
<tr>
<th></th>
<th>u</th>
<th>v</th>
<th>k</th>
<th>ε</th>
<th>c</th>
</tr>
</thead>
<tbody>
<tr>
<td>AB</td>
<td>(y/11)^{1/7} 0</td>
<td>0.02</td>
<td>0.1k^{3/2}</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>CD</td>
<td>(y/11)^{1/7} 0</td>
<td>0</td>
<td>∂k/∂x=0</td>
<td>∂ε/∂x=0</td>
<td>∂c/∂x=0</td>
</tr>
<tr>
<td>BC</td>
<td>1</td>
<td>0</td>
<td>∂k/∂y=0</td>
<td>∂ε/∂y=0</td>
<td>∂c/∂y=0</td>
</tr>
<tr>
<td>Solid surface</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>--†</td>
<td>∂c/∂t‡=0</td>
</tr>
<tr>
<td>Manure surface</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>--†</td>
<td>1.0</td>
</tr>
</tbody>
</table>

† No boundary condition needed.
‡ May be ∂c/∂x or ∂c/∂y depending on the location.
CHAPTER 6. COMPUTATIONAL PROCEDURE

Many algorithms can be used to solve the differential equations. For this study, the SIMPLER (semi-implicit-pressure-linked-equations-revised) algorithm presented by Patankar (1980) was used. SIMPLER as proposed by Patankar uses a control volume method and a staggered grid. Detailed discussion on this algorithm can be found in Patankar (1980).

6.1. Treatment of the Source Term

The $s_\phi$ term in Equation 5.22 is referred to as the source term. The treatment of the source term, as shown by Patankar (1980), can affect the convergence behavior of the problem. The source term is treated as two parts, a constant part and a part that changes with $\phi$. It can be expressed as,

\[ s_\phi = s_c + s_p \phi \]  
6.1

where,

\[ s_c = \text{constant portion of the source term} \]
\[ s_p = \phi \text{ dependent portion of the source term}. \]

As shown by Patankar (1980), the $s_p$ term should be zero or negative because of the stability concern. The same source term can be divided into different $s_c$ and $s_p$ terms depending on the stability situation. For flows that the numerical solution trends to diverge, a large $s_p$ value (absolute value) may help to get a converged solution. To make the problem
Table 5. The treatment of the source terms

<table>
<thead>
<tr>
<th>Equation</th>
<th>$S_\phi$</th>
<th>$S_p$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuity</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Momentum</td>
<td>$-\partial p/\partial x + \partial/\partial x_j (\Gamma_\text{eff} \partial u_j/\partial x_i)$</td>
<td>Part of the second term†</td>
</tr>
<tr>
<td>Turbulent energy</td>
<td>$\rho(G\varepsilon - \varepsilon)$</td>
<td>$-\rho\varepsilon/k$</td>
</tr>
<tr>
<td>Turbulent dissipation</td>
<td>$\rho(c_1\varepsilon G_{\varepsilon}/k - c_2^t\varepsilon^2/k)$</td>
<td>$-c_2^t\rho\varepsilon/k$</td>
</tr>
<tr>
<td>Odor concentration</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

† Part of the term that has velocity $u_i$ term in the discretized equation. † See Table 2.

more likely to converge, all the negative terms in the source term involving $\phi$ was put into the $s_p$ term. The $s_p$ used were shown in Table 5.

### 6.2. The Discretized Equation

To solve the partial differential equations, the equations need to be discretized. The discretization was carried out using the control volume approach and power-law scheme as described by Patankar (1980). One control volume (shaded area) and its surrounding grid points are shown in Figure 5. Integrating over the cell and using power-law scheme, the discretized equations for grid point $P$ can be written as,

$$ a_p\phi_p = a_E\phi_E + a_W\phi_W + a_N\phi_N + a_S\phi_S + b $$(6.2)

where,

$$ a_E = D_eA(|P_e|) + ||-F_e,0|| $$

$$ a_W = D_wA(|P_w|) + ||-F_w,0|| $$

$$ a_N = D_nA(|P_n|) + ||-F_n,0|| $$
\( a_s = DsA(|Ps|) + ||Fs|| \)

\( a_p = a_E + a_W + a_N + a_S + a_p^0 - s_p \Delta x \Delta y \)

\( b = s_d \Delta x \Delta y + a_p^0 \phi_p^0 \)

\( a_p^0 = \rho \Delta x \Delta y / \Delta t \)

\( \phi_p^0 = \) current \( \phi \) value (calculated from last iteration) at grid point \( p \)

\( D_e = \Gamma e \Delta y / (\delta x)_e \)

\( D_w = \Gamma w \Delta y / (\delta x)_w \)

\( D_n = \Gamma n \Delta x / (\delta y)_n \)

\( D_s = \Gamma s \Delta x / (\delta y)_s \)

\( F_e = (\rho u)_e \Delta y \)

\( F_w = (\rho u)_w \Delta y \)

\( F_n = (\rho v)_n \Delta x \)

\( F_s = (\rho v)_s \Delta x \)

\( A(|\eta|) = ||0, (1-0.1|\eta|)^5|| \)

\( P\eta = F\eta / D\eta \)

\(|a| = \) absolute value of \( a \)

\(||a,b|| = \) largest of \( a \) and \( b \)

\( \eta = e, w, n, s \)

The source terms, \( s_p \) and \( s_s \), were shown in Table 5 and were discretized by central difference.
Figure 5. Typical control volumes
6.3. The Solution Procedure

The SIMPLER scheme as proposed by Patankar (1980) uses staggered grids. The velocity calculation is based on a grid at the control volume surfaces and the other variables were based on the main grids to avoid the possibility of unrealistic velocity distributions to satisfy the continuity equation. Although the same discretized equation (Equation 6.2) is used for all the variables, for the same grid point, the P, N, S, E, W for u, v, and the scalar variables were different as shown in Figure 5.

The equations were solved iteratively. The basic idea is to guess a velocity field. Based on the guessed velocity, a pressure and new velocity field can be obtained. The new velocity field is the predicted velocity and is then corrected based on a pressure correction equation to make the procedure less explicit and the algorithm more likely to converge. The following is a brief description of the steps involved in the solution procedure for this study. For detailed discussion on the procedure, please refer to Patankar (1980). The summation in the discussion should be at the four neighboring locations.

1) Guess a velocity field.

2) Calculate the coefficients for the momentum equations and calculate the hat velocities, which will be used in pressure calculation equations.

\[ -\frac{u_e}{a_e} = \sum a_{nb} u_{nb} + b \]
\[ -\frac{v_n}{a_n} = \sum a_{nb} v_{nb} + b \]
3) Calculate the coefficients for the pressure equation and solve the pressure equation, which is derived from the continuity equation and can be expressed as,

\[ a_pP_p = a_EP_E + a_WP_W + a_NP_N + a_SP_S + b \]  

where,

\[ a_E = (pd)_e \Delta y \]
\[ a_W = (pd)_w \Delta y \]
\[ a_N = (pd)_n \Delta x \]
\[ a_S = (pd)_s \Delta x \]
\[ a_p = a_E + a_W + a_N + a_S \]
\[ b = [(\rho \bar{u})_w - (\rho \bar{u})_e] \Delta y + [(\rho \bar{v})_s - (\rho \bar{v})_n] \Delta x \]
\[ d_\eta = A_\eta / a_\eta \]
\[ A_\eta = \text{area at position } \eta. \]
\[ \eta = e, w, n, s \]

4) Treating this pressure field as the latest guess, solve the momentum equations to get the guessed velocity. The momentum equations solved are,

\[ a_u u^* = \sum a_{nb} u_{nb}^* + b + A_e (p_p^* - p_E^*) \]  

\[ a_v v^* = \sum a_{nb} v_{nb}^* + b + A_p (p_p^* - p_N^*) \]  

5) Solve the pressure-correction equation,

\[ a_pP_p = a_EP_E ' + a_WP_W ' + a_NP_N ' + a_SP_S + b \]  

The coefficients are the same as in Equation 6.4 except b, which is,

\[ b = [(\rho u^*)_w - (\rho u^*)_e] \Delta y + [(\rho v^*)_s - (\rho v^*)_n] \Delta x \]
6) Correct the velocity field using the velocity correction equations, which are

\[ u'_c = u'_c + u'_c' \]

\[ v'_n = v'_n + v'_n' \]

\[ a_e u'_c' = A_e(p'_p - p'_E) \]

\[ a_n u'_n' = A_n(p'_p - p'_w) \]

7) Solve the discretized equations for turbulence modeling.

8) Return to step 2) and repeat the steps until convergence criteria are reached. The flow field is now obtained.

9) Solve the conservation of the species equation to get the concentration distribution.

   Calculate the emission rate based on the calculated concentrations.

The Algebraic equations were solved using TDMA (Tri-Diagonal-Matrix-Algorithm) in a line by line sweeping (Patankar, 1980).

### 6.4. The Relaxing Coefficients

As shown in the last section, the problem was solved iteratively. Under relaxation was used in order to get a converged solution. In general, under-relaxation uses the following equation to calculate the value to be used for the next iteration cycle,

\[ \phi^{n+1} = (1 - \omega_\phi) \phi^{n+1} + \omega_\phi \phi^n \]

where,

\[ \phi^{n+1} = \text{value to be used for next iteration} \]
\( \phi^{n-1} \) = previously used value
\( \phi^n \) = currently calculated value
\( \omega_\phi \) = relaxing coefficient

For under-relaxing, the relaxing coefficient should be between 0 and 1. The relaxing coefficient used for different equations were shown in Table 6.

<table>
<thead>
<tr>
<th></th>
<th>u</th>
<th>v</th>
<th>k</th>
<th>( \varepsilon )</th>
<th>c</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.5</td>
<td>0.5</td>
<td>0.7</td>
<td>0.7</td>
<td>0.9</td>
</tr>
</tbody>
</table>

6.5. Convergence Criteria

The convergence criteria used for this study were,

Flow field:

\[ r_c < 2.0 \times 10^{-6} \]

where,

\( r_c = \) the residual of continuity equation, which is the \( b \) term in Equation 6.6

Conservation of species:

\[ \left( \frac{|c_{i,j}^{n+1} - c_{i,j}^n|}{c_{i,j}^n} \right)_{\text{max}} < 10^{-4} \]

where,

\( c_{i,j}^n = \) the odor concentration at grid point \( i,j \) at iteration number \( n \)
\[ c_{ij}^{n+1} = \text{the odor concentration at iteration } n+1 \text{ at grid point } i,j. \]

### 6.6. Grid Arrangement

As pointed out by Thangam (1991), proper grid resolution is important for the accuracy of numerical solution. Fine grid is needed at the solid surface region because of the sharp gradient at this region. For separation flow, fine grid reported was also needed at the separation point (Djilali, 1987). A non-uniform grid was necessary to ensure adequate grid resolution at the region that needs high grid resolution.

In this study, the case that demands the most grid points is when the tank is half full. It had the solid surface at the liquid level and ground level. The fine grids are also needed at the edge of the tank, where the separation takes place. The grid used in this study is shown in Table 7 and Figure 7.

The grid was first set up in the W/H=2 case with a half full tank in the following manner. First, the computation domain was divided into sections. Figure 6 shows the dividing up of the computation domain. A non-uniform grid was set up in a geometrical progress fashion for each section. Grid points were then added to the separation points and the solid surface region. The grid points in other regions were also adjusted to ensure the geometrical progression factor of less than 1.25 and the smooth transition between regions. This process continued until the change in emission rates and flow pattern between refinements were not significant. Based on this process, a grid of 152x139 was selected. Further refinement using a grid of 182x163 showed about 10% difference in emission rate and no noticeable difference...
Table 7. The grid arrangements

<table>
<thead>
<tr>
<th>section</th>
<th># of points</th>
<th>progression factor</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Full tank</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x1</td>
<td>31</td>
<td>D1.23</td>
</tr>
<tr>
<td>x2</td>
<td>40</td>
<td>A1.08</td>
</tr>
<tr>
<td>x3</td>
<td>40</td>
<td>D1.08</td>
</tr>
<tr>
<td>x4</td>
<td>41</td>
<td>A1.20</td>
</tr>
<tr>
<td>y1</td>
<td>36</td>
<td>A1.12</td>
</tr>
<tr>
<td>y2</td>
<td>25</td>
<td>D1.04</td>
</tr>
<tr>
<td>y3</td>
<td>78</td>
<td>A1.16</td>
</tr>
<tr>
<td><strong>Half full tank</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x1</td>
<td>31</td>
<td>D1.23</td>
</tr>
<tr>
<td>x2</td>
<td>40</td>
<td>A1.08</td>
</tr>
<tr>
<td>x3</td>
<td>40</td>
<td>D1.08</td>
</tr>
<tr>
<td>x4</td>
<td>41</td>
<td>A1.20</td>
</tr>
<tr>
<td>y1</td>
<td>36</td>
<td>A1.15</td>
</tr>
<tr>
<td>y2</td>
<td>45</td>
<td>A1.15</td>
</tr>
<tr>
<td>y3</td>
<td>20</td>
<td>D1.02</td>
</tr>
<tr>
<td>y4</td>
<td>38</td>
<td>A1.12</td>
</tr>
<tr>
<td><strong>Empty tank</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x1</td>
<td>31</td>
<td>D1.23</td>
</tr>
<tr>
<td>x2</td>
<td>40</td>
<td>A1.08</td>
</tr>
<tr>
<td>x3</td>
<td>40</td>
<td>D1.08</td>
</tr>
<tr>
<td>x4</td>
<td>41</td>
<td>A1.20</td>
</tr>
<tr>
<td>y1</td>
<td>70</td>
<td>A1.09</td>
</tr>
<tr>
<td>y2</td>
<td>30</td>
<td>D1.04</td>
</tr>
<tr>
<td>y3</td>
<td>38</td>
<td>A1.13</td>
</tr>
</tbody>
</table>

† Decreasing grid spacing
‡ Increasing grid spacing
Figure 6. Dividing the computation domain into sections
Figure 7. The grid arrangements

a. Full tank

b. Half full tank

c. Empty tank
in flow pattern. The 182x163 grid resulted in a grid spacing of less than 1/3 of grid 152x139 in the solid surface and separation point.

The same number of grid points (152x139) was also used in the case of full tank and the empty tank. Without the manure surface in the middle, the grid points were more concentrated in the ground level and also at the separation points for these cases, which would result in a better resolution than the case of a half full tank. Figure 7 shows the grids used for the half full tank, the full tank, and the empty tank cases for W/H=2.

For cases with W/H value other than 2, the difference is the width between the tank walls. It was assumed that this difference will not affect the grid solution requirement and the same geometrical progression factors were used.

For all cases, there were at least five grid points in the one equation model region on the manure surface and two grid points in the one equation model region for all the other solid surfaces.

6.7. Emission Rate Calculation

6.7.1. Emission rate calculation for the 2-D tank segment

The numerical simulation gives the velocity and other properties and also odor concentration at every grid point. Based on the calculated flow properties and calculated odor concentration distribution, the emission rates can be calculated. Since the air flow and the odor concentration at the manure surface were assumed at steady state, the emission rate is the transport rate of odor across any given cross section in the tank above the manure. The
transport of the odor is the sum of the convection and diffusion component. For one control
volume as shown in Figure 8, the dimensionless upward transport of odor across line AB can
be calculated as,

\[ q_{i,j}^* = \left[ \frac{v_{i,j}^* + v_{i,j+1}^*}{2} c_{i,j}^* + D_{df} \frac{c_{i,j-1}^* - c_{i,j+1}^*}{\Delta y_1^* + \Delta y_2^*} \right] \Delta x_i^* \]

Here the * is used to designate the dimensionless quantities. The dimensional upward
transport of odor across line AB can be calculated as,

\[ q_{i,j} = \left[ \frac{v_{i,j} + v_{i,j+1}}{2} c_{i,j} + D_{df} \frac{c_{i,j-1} - c_{i,j+1}}{\Delta y_1 + \Delta y_2} \right] \Delta x_i \]

Using the definition of the dimensionless quantities as defined in Equation 5.23, \( q_{i,j} \) can be
expressed as,

\[ q_{i,j} = q_{i,j}^* u_0 c_0 H \]

The odor transport for the cross section of the entire two-dimensional segment would be,

\[ Q = \sum_i q_{i,j} = u_0 c_0 H \sum_i q_{i,j}^* \]

The summation in Equation 6.13 should be the grid points across the tank.

Odor transport was calculated at the middle section of the tank above the manure surface
according to Equation 6.13. Different sections were also calculated to check if there were
any differences between the sections. The differences between the sections was very small
(<0.5%). For the full tank cases, difficulty existed as how to calculate the emission rate with
reasonable effect if the tank is 100% full. To simplify the problem, the tank was kept 99.9% full and the emission was then calculated the same way as the other cases.

6.7.2. Emission flux calculation for the 2-D tank segments

Emission rate calculation based on Equation 6.13 is a function of the surface area and a comparison between segments with different surface areas may not be meaningful. The emission flux, which is the emission rate per unit area, is better suited for this purpose. According to the definition, the odor flux, \( F \), can be calculated as
\[ F = \frac{Q}{A} \quad 6.14 \]

Substitute Equation 6.13 into 6.14, and using the relationship that \( A = A^*H \)

\[
F = \frac{u_0 c_0 H \sum q_{i,j}^*}{HA^*} \\
= \frac{u_0 c_0 \sum q_{i,j}^*}{A^*} \\
= u_0 c_0 F^* \quad 6.15
\]

where,

\[ A^* = W^*, \text{ which is } 0.6, 1.2, 1.8, 2 \text{ for } W/H=0.6, W/H=1.2, W/H=1.8, \text{ and } W/H=2 \text{ respectively.} \]

### 6.7.3. Emission flux for the tank

To estimate the odor flux from the round tank, the fluxes calculated according to Equation 6.15 for different two-dimensional segments (Figure 3) were weighted by the segment area. The odor flux from the tank was then calculated as,

\[ F_{\text{tank}} = \sum_{i=1}^{4} R_i F_i \quad 6.16 \]

where,

\[ R_i = \text{ the fraction of the tank area for segment } i. \]

\[ F_i = \text{ The emission flux from segment } i. \]
6.7.4. Wind speed correction

The wind speed, $u_w$, is usually measured at 10 m above the ground. The characteristic velocity used for this study, $u_0$ is defined at the top of the computation domain, which is 11 times the tank height, H. It is more convenient to use the actual wind speed in practical application. Using the $1/7$ power wind profile, $u_0$ can be expressed in terms of $u_w$ as

$$\frac{u_0}{u_w} = \left(\frac{11H}{10}\right)^{\frac{1}{7}} \approx H^{\frac{1}{7}}$$ 6.17

Substitute Equation 6.17 into Equation 6.15, the flux from the two dimensional tank can be calculated as,

$$F = u_w c_0 H^{\frac{1}{7}} F^*$$ 6.18

6.7. Computer Equipment

The calculations were carried out on DEC 3000 from Digital Equipment Corporation. One run usually took about 5000 iterations to converge, which is four hours of CPU time.
CHAPTER 7. EXPERIMENTAL ARRANGEMENT

To verify the numerical prediction, an experimental study was needed. Because of the size of the tank and the unsteady nature of the wind, an onsite experimental study was not possible. A wind tunnel study on a scaled model was carried out to verify the numerical prediction.

7.1. Wind Tunnel Setup

A wind tunnel was constructed for this study. The wind tunnel is shown in Figure 9. The air flow was driven by two 1.22 m (48") fans placed in parallel. The fans were identical belt driven fans made by Hired Hand Inc., Iowa. The motor is 1.2 kw. The fans were set on a separate frame of the test section to isolate the possible vibration affect on the test section.

The test section dimensions were determined by the scaled model dimensions and available dimension of the building material. The commercially available plywood, which was used as the construction material, comes with 1.22m x 2.44m (4’x8’) in dimensions and the commonly available lumber with the maximum length of 3.66m (12’). The gross dimensions for the test section were determined to be 3.66m x 2.44m x 1.22m (12’x8’x4’, length x width x height). The four sides (top, bottom, and two side walls) were constructed separately. They were held together by five steel rods on each side. Weather stripping was placed on the joint surfaces to prevent leakage. The four corners were held to 90°±2°. The width and height of the test section were measured at the entrance, exit, and at the middle.
Figure 9. Wind tunnel setup (unit is in meter)
The width of the test section was measured as 2.22 m ± 0.013 m (87.5" ± 0.5") and the height was 1.23 m ± 0.005 m (48.3" ± 0.2"). The length of the test section was measured as 2.90 m.

The two-dimensional tank segment was simulated by two plates. The scaled tank height was 0.15 m (6"). The plates were placed on the side wall. The plate gave a wind tunnel blockage of 6%. As shown by Hunt (1982), this blockage would lead to error in mean properties of less than 2%. Bottcher (1985) used a wind tunnel to study natural ventilation in buildings and used a model with blockage of 7%.

To get a uniform inflow and reduce the turbulence in the inflow, five layers of screens made of fiberglass of 18x16 mesh were placed at the tunnel entrance as the flow straightens. They were placed 0.08 m apart. A board with holes about 1 cm in diameter and 2 cm apart was placed at the exit of the test section to isolate the flow in the test section from the possible changes in the outside. Two plexiglass plates of 0.91 m x 1.22 m (3' x 4') were placed on the same side of the side wall to provide the possibility of lighting and visualization of the flow field. Two other plexiglass plates of 0.30 m x 1.22 m (1' x 4') were placed on the ceiling of the test section for the same purpose. A trap door was constructed in the test section near the exit to allow access. The trap door was screwed to the tunnel from outside with a gasket seal to prevent leakage.

7.2. The Flow Field Entering the Test Section

The flow field entering the test section was measured to make sure the flow was acceptable in uniformity. The velocities at the selected points, as shown in Figure 10, were
Figure 10. Points where the velocity was measured

Figure 11. The velocity profile along line AB in Figure 10
measured at the section that is 5 cm from the last layer of flow straightener. The points measured are marked by x and are alone a horizontal line, a vertical line, and two diagonal lines. A hot-film velocity transducer of Model 8470-50M-V from TSI, Inc., St. Paul, MN. was used for the measurement. The transducer has a designed range of 0 - 5.00 m/s and has the accuracy of ±3% of reading and ±1% full range. The velocity was measured for a 1 minute duration at 0.5 Hz. The results were averaged as the mean velocity for the point. The results showed an average velocity of 1.39 m/s, which is relatively low speed. The velocities ranged from 1.30 m/s to 1.48 m/s with standard deviation of 0.042 m/s. The velocity data are listed in Appendix A. Figure 11 shows the velocity profile along the horizontal line AB as shown in Figure 10. The low velocity in the test section is due to the lack of inlet venturi and also the friction loss from the isolation board at exit.

7.3. The Scaled Model of the Simplified Tank Segments

The flow in the three dimensional tank was simplified to four two dimensional flows. The simplified tank segment was represented by two identical plexiglass plates. The scaled tank height is 0.15 m (6"). The liquid surface was also represented by a plexiglass plate, which can be inserted into the notches in the tank plates. The different W/Hs were accomplished by using different surface plates and different manure depths which were simulated by placing the surface plates at different locations in the tank plates.
7.4. The Flow Pattern Visualization and Air velocity measurement

Flow patterns in the wind tunnel were observed using smoke generated by smoke candler (type 10-60, E. Vernon Hill, Inc, CA.). Lighting was provided by a lighting box, shown in Figure 12. The light box was designed to provide a narrow light beam to get a better view of the flow pattern for selected two-dimensional planes. Flow patterns for W/H=2 for three manure depths of h/H=0, h/H=0.5, h/H=1 were run to verify the numerically predicted flow pattern. The flow pattern was photographed and also video taped. However, the flow pattern inside the tank was not clear enough on the photo because of the low air velocity inside the tank. The flow pattern was drawn based on the video tape replay.

To verify the predicted air velocity inside the tank, the air velocities were also measured along the center of the tank on W/H=2 and h/H=0 case.

It would be beneficial to be able to verify the predicted emission rate. However, no adequate way was found to measure the emission rate. The same difficulty in measuring the emission rate discussed in the discussion of why the similitude study was not used also existed.
Figure 12. A light box for flow visualization
CHAPTER 8. RESULTS AND DISCUSSION

8.1. The Verification of the Numerical Prediction

A major part of this study is a numerical study of the air flow and odor emission rate from a manure storage tank. To verify the validity of the predicted results, wind tunnel tests were carried out. However, the wind tunnel test was not designed as a similitude study because of the difficulties involved with a similitude study for this problem. Thus, separate numerical simulations were carried out according to the wind tunnel setup in addition to numerical simulation of the two-dimensional tank segments. It was assumed that if the numerical simulation can predict the experimental study in the wind tunnel, it can also predict the flow in a full scale tank.

8.1.1. The flow pattern verification

The flow pattern verification was done for W/H=2 with a full, half full, and an empty tank. In the wind tunnel study, the scaled tank height was 0.15 m (6") and the wind speed was 1.39 m/s. The incoming velocity profile is shown in Figure 7, which was basically uniform with a slightly higher velocity near the wall.

Separate numerical simulations were performed based on the above conditions. Compared with the parameters specified for a full size tank, the differences here included:

*The incoming velocity profile*: A uniform profile was specified here and 1/7 power law for a full sized tank.
The tank size and wind speed: The actual dimensions and wind speeds were used to calculate the Reynolds number, which affects the flow.

The turbulence intensity in the incoming air flow: 7% was used because of the flow straighter and 15% was used in the full sized tank.

Exit velocity profile: A uniform profile was specified and 1/7 power law for the full sized tank.

8.1.1.1. The case of tank full

The results of numerical simulation and experimental flow pattern are shown in Figure 13. As shown, both numerical predictions and experimental results showed that the flow in the tank was dominated by the recirculation flow behind the tank. The separation happened at the leading edge of the tank and no reattachment of the main flow on the manure surface was observed. Both numerical prediction and experimental results showed a larger recirculating region behind the tank. At a relatively large distance down stream (about 10 times the tank height) the height of the recirculating region was almost parallel to the ground. Overall, little difference in flow patterns were observed between the numerical prediction and experimental results except some unsteadiness observed on the video tape replay.

8.1.1.2. The case of half full tank

The results of numerical simulation and experimental flow pattern for the case of a half full tank are shown in Figure 14. As shown in the figure, both numerical prediction and
experimental results showed that the air flow in the tank was originated from the recirculating flow behind the tank. The recirculation flow touched the manure surface and split into two portions and formed two rotary zones rotating in two different directions. The separation happened at the leading edge of the tank. A similar flow pattern behind the tank as the case of the full tank was observed. Little difference were observed between the experimental results and numerical prediction except some unsteadiness observed on the video tape replay.

8.1.1.3. The case of empty tank

The results of numerical simulation and experimental flow pattern for the case of empty tank are shown in Figure 15. As shown, both numerical prediction and experimental results showed that the air flow in the tank originated from the recirculating flow behind the tank. The majority of the air entering the tank was entrained into the main flow without forming a clear dead air zone. The air flow at the liquid surface region was almost still in both numerical and experimental results. A similar flow pattern behind the full tank was observed. Generally, the flow pattern prediction agreed with the experiment.

The air flow in the experiment in the near liquid surface region was unstable and almost still from the video tape replay but showed the tendency of moving in the direction of the main flow. This was not observed in the numerical simulation. This discrepancy may be due to the incoming velocity profile difference between the experiment and the numerical simulation (Figure 11). The incoming velocity in the experiment was not uniform and was slightly higher near the wall region, which would result in differences in
Figure 13. The predicted and experimental flow pattern for a full tank ($W/H=2$, $h/H=1$)
Figure 14. The predicted and experimental flow pattern for a half full tank
(W/H=2, h/H=0.5)
Figure 15. The predicted and experimental flow pattern for an empty tank (W/H=2, h/H=0)
flow pattern. It is also possible that the difference was due to the turbulence model or other
umerical error. Because of the fact that the difference existed in the region that the air was
almost still, no further study was done to pinpoint what may be the exact cause of the
difference

8.1.2. The air velocity profile

The magnitude of the air velocity in the empty tank and W/H = 2 case was measured
along the center line of the simplified tank to verify the numerical prediction. This case was
selected because it was the most difficult to predict from the flow pattern verification.

The results were shown in Figure 16 with the results from the numerical prediction. The
maximum velocity measured was 1.91 m/s, which agreed with the numerical prediction of
2.01 m/s. Figures 16 shows that the prediction also agreed well with the experiment result in
terms of the magnitude of velocity.

The results of experimental verification showed that the numerical prediction agrees with
the experimental results in flow patterns and also in the magnitude of the velocity. From
these results, it can be concluded that the numerical scheme and turbulence model are suited
for the study with reasonable accuracy and they are not likely to be the major source of error
in the numerical prediction.
Figure 16. The air velocities along the center of the tank \((W/H=2, h/H=0)\)
8.2. The Air Flow Pattern in the Full Size Tank

In this study, one three-dimensional flow in the open tank problem was simplified to four two-dimensional problems. Three different manure depths were simulated for every two-dimensional case, empty (but the bottom covered with manure), half full, and full. Thus a total of 12 cases were simulated (not include the simulations for verification). The predicted velocity field and velocity contour at the tank region for case W/H=2 and the half full tank are shown in Figure 17. Figure 18 shows the entire computation domain. The velocity field in the tank region for case W/H=1.2, and W/H=0.6 and for all three manure levels and the case of W/H=2 for the empty tank and the full tank were shown in the figures listed in the Appendix because of the large number of figures. The case for W/H=1.6 was similar to W/H=2 and the velocity and concentration field was not shown. Because of the storage space available on the computer system, only the emission rate data were saved for the case.

The flow patterns showed that the flow inside the tank was driven by air entering and leaving the tank at the edges of the tank.

The lengths of the recirculation zone behind the tank for all the cases, which was defined as the length from the leading tank wall to the position where the horizontal velocity component of the air flow at ground level started to be in the main flow direction, were tabulated in Table 8. The recirculation zones had a size of 7.1 to 10.4 times of the tank height, which was in agreement with the observation of Schofield (1990). He found the reattachment lengths from 7-14 times the obstacle height for two-dimensional obstacles after reviewing a number of studies.
Figure 17. The flow pattern in the tank region (W/H=2, h/H=0.5)
Figure 18. The velocity contour in the tank region (W/H=2, h/H=0.5)
Figure 19. The odor concentration contour in the tank region (W/H=2, h/H=0.5)
Figure 20. The flow pattern in the entire domain (W/H=2, h/H=0.5)
Figure 21. The odor concentration contour in the entire domain (W/H=2, h/H=0.5)
Table 8. The predicted recirculation lengths

<table>
<thead>
<tr>
<th></th>
<th>W/H=0.6</th>
<th>W/H=1.2</th>
<th>W/H=1.6</th>
<th>W/H=2</th>
</tr>
</thead>
<tbody>
<tr>
<td>h/H=0</td>
<td>8.9</td>
<td>7.8</td>
<td>--‡</td>
<td>7.1</td>
</tr>
<tr>
<td>h/H=0.5</td>
<td>10.4</td>
<td>9.1</td>
<td>--</td>
<td>8.1</td>
</tr>
<tr>
<td>h/H=1</td>
<td>8.9</td>
<td>7.8</td>
<td>--</td>
<td>7.1</td>
</tr>
</tbody>
</table>

‡ Not evaluated

8.3. The Concentration Distribution

The predicted odor concentration distribution at the tank region and for the entire computation domain for case W/H=2 and the half full tank are shown in contour form in Figure 19. Case W/H=1.2, and W/H=0.6 and for all three manure levels and the case of W/H=2 for the empty tank and the full tank are shown in the figures listed in the Appendix because of the large number of figures. The case for W/H=1.6 was similar to W/H=2 and was not shown.

The concentration distribution showed a very sharp concentration gradient in the manure surface region, indicating the importance of the viscous sublayer. The odor concentration leaving the tank was relatively low in concentration (at about 1% to 5% level of the concentration at the manure surface). A slow reduction in odor concentration was observed after leaving the tank.
8.4. The Calculated Odor Emission Rates

8.4.1. The calculated results

The odor emission fluxes for every two-dimensional tank segments were calculated according to Equation 6.18 and the odor flux for the simplified tank was calculated according to Equation 6.16. The results were listed in Table 9.

Table 9. Calculated Odor fluxes (times $u_w c_0 H^{1/7} \times 10^{-3}$ ou*m/m/s)

<table>
<thead>
<tr>
<th></th>
<th>W/H=0.6</th>
<th>W/H=1.2</th>
<th>W/H=1.6</th>
<th>W/H=2</th>
<th>Tank</th>
</tr>
</thead>
<tbody>
<tr>
<td>h/H=0</td>
<td>0.18</td>
<td>0.78</td>
<td>1.2</td>
<td>1.3</td>
<td>1.2</td>
</tr>
<tr>
<td>h/H=0.5</td>
<td>0.65</td>
<td>1.2</td>
<td>1.3</td>
<td>1.3</td>
<td>1.3</td>
</tr>
<tr>
<td>h/H=1</td>
<td>2.4</td>
<td>2.3</td>
<td>2.3</td>
<td>2.3</td>
<td>2.3</td>
</tr>
</tbody>
</table>

8.4.2. Comparison with the existing data

As shown in the literature review, only three studies, done by Li et al. (1994), Bode (1991), and Carney and Dodd (1989) were found involving odor emission rate. The study of Carney and Dodd (1989) could not be used because it did not specify where to measure the odor concentration. Although the other two studies would not provide accurate enough results to judge the accuracy of this study, they were actual onsite measurements of manure storage tanks and provide a check on the magnitude of odor emission rate from the tank.

Li et al. (1994) estimated the odor emission rate from a manure storage tank of 31m x 7.6m (diameter x height) based on the field measurement of the odor plume width at different distances from the tank using the Gaussian plume model. The tank had a diameter/height of approximately 4 and the manure depth ranged from about full to near empty. The estimated
emission rates ranged from $1.5 \times 10^4$ to $7.5 \times 10^5$ ou.m$^3$/s. The lowest occurred during the middle of the night at wind speeds of 4.4 m/s. The highest during the noon hour at wind speed of 7.3 m/s. The odor concentration at the manure surface was measured to be greater than 1200 ou. The exact concentration was not determined because of the limitation of the instrument used. If the concentration at the manure surface is assumed at 5,000 ou and the tank assumed to be at half full using the case of $W/H = 2$ and wind speed of 7 m/s, the odor flux from Table 8 would be $1.4u_0 c_0 H^{1/7} \times 10^{-3}$ ou$^*$$m^3/(m^2s)$ from the tank. The odor emission rate using the result of this study would be,

\[
Q = FA
\]

\[
= 1.4\times u_0 c_0 H^{1/7} \times 10^{-3} \pi \times 31^2 / 4
\]

\[
= 5 \times 10^4 \text{ ou}^* \text{m}^3/\text{s}
\]

which is comparable to the results of Li et al. of $1.5 \times 10^4$ to $7.5 \times 10^5$ ou.m$^3$/s.

Bode (1991) studied the odor and ammonia emission from tanks of 2m x 1.9m (diameter x height). A box with a fan providing air flow of 48 m$^3$/min was put on top of the tank. The exhaust was measured for odor concentration. For pig manure, the odor concentration of 120 to 200 ou were measured in the exhaust. Assuming an exhaust concentration of 160 ou, the odor emission from the tank would be,

\[
Q_{\text{bode}} = 160 \text{ ou} \times 48 \text{ m}^3/\text{min} \times 1 \text{ min/60 sec}
\]

\[
= 128 \text{ ou} \times \text{ m}^3/\text{s}
\]
Assuming the air flow is equivalent to 4 m/s wind speed and the odor concentration on the manure surface was 5,000 ou, and the tank was half full, and the odor emission rate using the result of this study would be,

\[ Q = FA = 1.2 \cdot u_w c_0 H^{1/7} \cdot 10^{-3} \cdot \pi^{2/4} = 100 \text{ ou} \cdot \text{m}^3/\text{s} \]

The results are also in the same magnitude of this study.

8.5. Application to Other Conditions

The tank simulated has a diameter to height ratio of 2. Only one Reynolds number was simulated (Re=2.7x10^6). To find out the sensitivity of emission rate to Reynolds number, Reynolds numbers of 1x10^6, 3x10^5, 1x10^5, 3x10^4, and 1.4x10^4 were simulated for the case of W/H=2 and h/H=0.5 in addition to Reynolds number of 2.7x10^6. Different Reynolds numbers were achieved by using different wind speeds or tank height, which are equivalent for non-dimensionalized equations. Higher Reynolds number was not simulated because it would require a denser grid.

The flow patterns and velocity contours of Reynolds number of 3x10^5 and 1.4x10^4 are shown in Figure 22 to Figure 25 respectively. Compared with Figure 17 and 18, which is the flow pattern and velocity contour for Reynolds number of 2.7x10^6, no noticeable difference was found in the flow pattern. The velocity contour showed some differences between different Reynolds numbers.
Figure 22. The air flow pattern at $Re=3\times10^5$ ($W/H=2$, $h/H=0.5$)
Figure 23. The air velocity contour at Re=3x10^5 (W/H=2, h/H=0.5)
Figure 24. The air flow pattern at $Re=1.4\times10^4$ ($W/H=2$, $h/H=0.5$)
Figure 25. The velocity contour at $Re=1.4 \times 10^4$ ($W/H=2$, $h/H=0.5$)
The non-dimensional emission fluxes calculated using different Reynolds numbers were listed in Table 10. The result showed higher non-dimensional emission fluxes with the lower Reynolds number. But the change is relatively small for a large range of Reynolds number. The change was less than 80% when Reynolds number changed from $2.7 \times 10^6$ to $1 \times 10^5$. Depending on the accuracy of the requirement, the results of this study can be used for a wide range of Reynolds numbers.

The conclusion that the non-dimensional emission flux is higher at lower Reynolds number showed that the non-dimensional emission flux is higher at lower velocity. However, the actual dimensional emission flux is actually lower because the non-dimensional emission rate is multiplied by $u_0 c_0$ to convert it to dimensional emission rate. The conclusion of higher emission rate at lower velocity (lead to lower Reynolds Number) should not be drawn.

8.6. The Impact of W/H for 2-D Tank Segments

In the W/H range studied (0.6-2), the results in Table 9 showed a strong link between the emission rate and W/H for the two-dimensional segments except the case were the tank was

<table>
<thead>
<tr>
<th>Table 10. Odor emission fluxes at different Reynolds numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7x10^6</td>
</tr>
<tr>
<td>----------</td>
</tr>
<tr>
<td>1.3†</td>
</tr>
</tbody>
</table>

† times $u_0 c_0 H^{1/7} * 10^{-3}$ to get the dimensional fluxes
full. As shown in the flow patterns in the Appendix, small W/H hindered the circulation movement of air in the tank, thus greatly reduced the emission rate. When the tank was full, increase in W/H actually had slightly smaller emission flux, as shown in Table 9. The odor concentration of the air in contact with the manure surface is higher with the increase of W/H as air picks up the odor upstream. This reduces the concentration gradient and the emission rate.

The results listed in Table 9 indicated that diameter to height ratio is an important factor affecting the emission rate. For different tanks with diameter to height ratio other than 2, the result can only be used as a reference and further study is needed to determine the emission rate.

8.7. The Impact of Manure Depth

Table 9 shows the odor fluxes calculated for each two-dimensional tank segment and for the tank at different manure depth. Generally, the emission rate was higher with the higher manure level for each two dimensional segment with the exception of W/H=2, which showed the same in emission rate for h/H=0 and h/H=0.5. The emission rate changes for different manure depths which can be explained by the flow pattern differences for different manure depths. Generally, the higher manure depth resulted in higher velocity at the manure surface which resulted in an increase in the odor emission rate. For the case of W/H=2, the manure depth of h/H=0.5 resulted in a slightly higher velocity on part of the manure surface, but it
also shifted the rotary zone to the lee ward side of the tank, leaving part of the manure surface with low air velocity compared with the case of h/H=0.

As shown in Table 9, the odor emission rates for manure depth 0 and 0.5 H did not show a significant difference for the tank but the emission rate was higher when the tank was full. These results showed that the manure depth may not be important unless the tank is about full.

8.8. The Uncertainties

8.8.1. Diffusion Coefficients

The emission rate is directly related to diffusion coefficient $D_{\text{eff}}$ ($D_{\text{eff}} = \nu/\text{Sc} + v_t/\sigma_c$). Two parameters control the diffusion coefficient: the Schmidt number (Sc), which controls the laminar portion of the coefficient, and $\sigma_c$, which controls the turbulent portion of the coefficient. The $\sigma_c$ value is more likely to be dependent on the property of the flow and less dependent of the species in question and thus not likely to be a source of error. The Schmidt number is the property of the species and the media.

The Schmidt number used for this study of $\text{Sc} = 1.0$ was an estimate and probably are not the actual values. The sensitivity of emission rate to Sc was checked by calculating the emission rate at $\text{Sc}=0.5$ and 2.0. The results showed Sc of 0.5 gave an emission of 50% higher than that of Sc =1. Sc of 2 gave an emission rate which was 35% lower than that of Sc=1. If a better knowledge of Sc number for odor in air is available, the odor emission flux can be estimated from the above results.
8.8.2. Two-dimensional simplification

The three-dimensional flow in the open manure storage tank was simplified to four two-dimensional segments. This may underestimate the emission rate because two-dimensional simplification forces more air to go over the tank and creates a larger recirculation zone. The three-dimensional flow is more likely to reattach and flow into the tank. The exact amount of error due to two-dimensional simplification is unknown at this point.

8.9. Using the Result in Practical Application

If the results of this study as listed in Table 9 is to be used for practical application, the measurement of odor concentration at the manure surface and wind speed should be measured to get more accurate results. As shown in the concentration contours, the concentration gradient near the surface is extremely high and sampling with a tube near the surface will probably underestimate the concentration on the surface significantly. The concentration is probably close to the saturation concentration on the surface and should be measured accordingly.

The wind speed should be measured at 10 meters above the ground. Corrections should be made if the speed is not measured at that level.
CHAPTER 9. CONCLUSION

This research was a study of the odor emission rate from the manure storage tank. The approach of using the concentration at the manure surface concentration, the tank dimensions, and wind speed to calculate the emission rate avoided the need of knowing the state of the manure and the bio-process taking place in the manure. The numerical simulation using SIMPLER algorithm and a two layer turbulence model with a grid of 159x139 was used.

The predicted emission rate agreed with the field measurement results available, indicating the predicted emission rate is as accurate as the data that is available. More reliable experimental results are needed to see if the numerical results are more accurate. A wind tunnel was constructed for this study. Experiment verification showed that the flow pattern and velocity profile prediction were in agreement with the experimental results. The numerical simulation showed how different factors affect the emission rate. The information may be used as a guide for future analytical or experimental studies.

The calculated odor flux was a function of many factors. It was found to be in the order of $10^3 c_0 u_s H^{1/7}$. The Reynolds number sensitivity test showed that the result can be used for a range of Reynolds numbers with acceptable error.

When using this result, the correct measurement of odor concentration at the manure surface should be measuring its saturation concentration. Because of the sharp gradient at the
surface, measuring the concentration near the surface will significantly under estimate the concentration.
This study focused on a tank with a diameter to height ratio of 2. The flow in the tank was also simplified from three dimensional to two-dimensional with limited experimental verification. Based on these limitations, the further research direction to improve upon and build on this study may include:

1. Study on the possibility of partial covering to reduce the odor emission from the tank. The flow pattern shown in this study suggested that it is possible to partially cover the tank and reduce the odor emission significantly.

2. Similar research to this study on odor and other gaseous emission for a different tank configuration, lagoon, and earth storage. Although numerical study is not perfect, a carefully carried out numerical study can certainly meet the accuracy requirements of odor emission studies.

3. Improve upon the numerical simulation used in this study to better resemble the flow pattern of three-dimensional flow. Possibly, the incoming velocity profile can be changed to better reflect the amount of air forced over the tank.

4. Experimental study that can provide adequate experimental data to verify the emission rate model. A similitude study of an open tank is going to be difficult because of the difficulties discussed in this research. Experiments may be possible strictly to verify the numerical prediction.
5. Three-dimensional modeling. If the computer resource allow, expand the study to three-dimensional would eliminate the error of simplifying the three-dimensional flow to two-dimensional.

6. Complete modeling of the entire generation and emission process, including the process inside the manure, which would enhance the understanding of the odor emission process greatly.
REFERENCES


Committee on odors from stationary and mobil sources. 1979. Odors from stationary and mobile sources. National academy of sciences, Washington D. C.


Han, T. 1989. Computational analysis of three-dimensional turbulent flow around a bluff body in ground proximity. AIAA Journal. 27(9): 1213-1219.


23: 79-98.

21: 205-34.

Smith, R. J. 1993. Dispersion of odours from ground level agricultural sources. J. agric.
Engng Res. 54: 187-200.

Snyder, William H. 1985. Fluid modeling of pollutant transport and diffusion in stably

Snyder, W. H. 1972. Similarity criteria for the application of fluid models to the study of air


Journal. 29: 2069-2076.


Solberg, T. and K. J. Eidsvik. 1989. Flow over a cylinder at a plane boundary-a model based

involving both first and second derivatives. International Journal for Numerical Methods
in Engineering. 4: 551-559.

for near wall turbulence. AIAA paper. 90-1481.

Diego.


ACKNOWLEDGMENTS

The author would like to thank my advisor, Dr. Dwaine Bundy, for guiding me through this project and providing me the opportunity to work on a project of great interest to me. The six years that I have spent under his guidance for my M.S. and Ph.D. degrees have been very rewarding and professionally fulfilling.

Dr. Steven Hoff provided me with invaluable help and support throughout this project. His dedication and enthusiasm to research and helping students set the standard for me to follow in my career.

I would also like to thank Dr. Earl Hammond, Dr. Ron Nelson, and Dr. Stewart Melvin for their encouragement and suggestions, which helped me successfully complete this project.

Marc Lott, the technician in the department, provided me with many creative ideas to solve my problems during the experiment period. Mrs. Delores Vandepol proof read this thesis and many other items the author wrote. Their help is greatly appreciated.

Thanks are also extended to all those who helped me at Iowa State University.

Finally, I want to thank my wife Lirong and son Bill for their understanding, love, and support throughout this project. It is their support and sacrifices that made it possible for me to concentrate on this project. I dedicate this work to them.
APPENDIX A.
Table A.1. The velocities measured at the wind tunnel test section entrance

<table>
<thead>
<tr>
<th>point</th>
<th>velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.46</td>
</tr>
<tr>
<td>2</td>
<td>1.44</td>
</tr>
<tr>
<td>3</td>
<td>1.44</td>
</tr>
<tr>
<td>4</td>
<td>1.41</td>
</tr>
<tr>
<td>5</td>
<td>1.41</td>
</tr>
<tr>
<td>6</td>
<td>1.37</td>
</tr>
<tr>
<td>7</td>
<td>1.36</td>
</tr>
<tr>
<td>8</td>
<td>1.37</td>
</tr>
<tr>
<td>9</td>
<td>1.37</td>
</tr>
<tr>
<td>10</td>
<td>1.39</td>
</tr>
<tr>
<td>11</td>
<td>1.40</td>
</tr>
<tr>
<td>12</td>
<td>1.40</td>
</tr>
<tr>
<td>13</td>
<td>1.38</td>
</tr>
<tr>
<td>14</td>
<td>1.38</td>
</tr>
<tr>
<td>15</td>
<td>1.38</td>
</tr>
<tr>
<td>16</td>
<td>1.46</td>
</tr>
<tr>
<td>17</td>
<td>1.48</td>
</tr>
<tr>
<td>18</td>
<td>1.39</td>
</tr>
<tr>
<td>19</td>
<td>1.42</td>
</tr>
<tr>
<td>20</td>
<td>1.40</td>
</tr>
<tr>
<td>21</td>
<td>1.38</td>
</tr>
<tr>
<td>22</td>
<td>1.32</td>
</tr>
<tr>
<td>23</td>
<td>1.38</td>
</tr>
<tr>
<td>24</td>
<td>1.38</td>
</tr>
<tr>
<td>25</td>
<td>1.34</td>
</tr>
<tr>
<td>26</td>
<td>1.40</td>
</tr>
<tr>
<td>27</td>
<td>1.29</td>
</tr>
<tr>
<td>28</td>
<td>1.31</td>
</tr>
<tr>
<td>29</td>
<td>1.39</td>
</tr>
<tr>
<td>30</td>
<td>1.39</td>
</tr>
<tr>
<td>31</td>
<td>1.35</td>
</tr>
<tr>
<td>32</td>
<td>1.36</td>
</tr>
<tr>
<td>33</td>
<td>1.38</td>
</tr>
<tr>
<td>34</td>
<td>1.34</td>
</tr>
<tr>
<td>35</td>
<td>1.35</td>
</tr>
<tr>
<td>36</td>
<td>1.35</td>
</tr>
</tbody>
</table>

† See Figure 10.
Figure B.1. The odor concentration contour and velocity field (W/H=2, h/H=0)
a. Flow pattern

b. Concentration contour

Figure B.2. The odor concentration contour and velocity field (W/H=2, h/H=1)
Figure B.3. The odor concentration contour and velocity field (W/H=1.2, h/H=0)
Figure B.4. The odor concentration contour and velocity field (W/H=1.2, h/H=0.5)
Figure B.5. The odor concentration contour and velocity field (W/H=1.2, h/H=1)
Figure B.6. The odor concentration contour and velocity field (W/H=0.6, h/H=0)

Legend:
- \(1.000 \times 10^0\) (ref.)
- \(1.000 \times 10^{-1}\)
- \(5.000 \times 10^{-2}\)
- \(1.000 \times 10^{-3}\)
- \(5.000 \times 10^{-4}\)
- \(1.000 \times 10^{-5}\)
- \(5.000 \times 10^{-6}\)
- \(1.000 \times 10^{-7}\)
- \(0.000 \times 10^{-8}\)
Figure B.7. The odor concentration contour and velocity field (W/H=0.6, h/H=0.5)
b. Concentration contour

Figure B.8. The odor concentration contour and velocity field (W/H=1.2, h/H=1)