Numerical study on flow and pollutant dispersion inside street canyons

Yunkai Yang
杨云铠

KTH Royal Institute of Technology
School of Architecture and the Built Environment
Department of Civil and Architectural Engineering
Division of Building Service and Energy Systems

Licentiate thesis
Abstract

This thesis analyzes the characteristics of flow pattern and vehicle-emitted pollutant dispersion in roughness surface layer. In an urban environment, wind flow and transported-pollutant source interfere strongly with buildings and other roughness elements on the surface ground, which results in complex characteristics of flow pattern and pollutant dispersion in 3D circumstances. The present study intends to simplify the research domain and investigate the fundamental modeling problems that exist in the field. The current physical research topic is restricted to 2D street canyon in equilibrium conditions. The study is motivated by the fact that characteristics of flow pattern and pollutant distribution inside street canyons are important for public health. The research has applied the computational fluid dynamics (CFD) methodology. To date, insights have typically focused on idealized street canyons without strictly limited boundary conditions and turbulence models. Those approaches face challenges related to their applicability to real urban scenarios or the reliability of prediction results.

The thesis examines the influence of grid density, turbulence models and turbulent Schmidt number on pollutant distribution at windward and leeward surfaces of street canyon. Since numerical results usually are validated with wind-tunnel measurement data, the results between full-size model and wind-tunnel model are compared in order to test the Reynolds number effect. The lack of measurement data means that the morphometric method is used to generate upcoming wind profile, including the mean vertical velocity and turbulence parameters. The thesis also analyzes the potential errors brought by the method (Scenario A).

Based on the evaluated numerical model, the thesis continues to study the impacts of surrounding buildings and geometry of street canyon on flow and pollutant distribution inside street canyons. The effect of wind on pollutant distribution inside street canyons was also investigated (Scenario A). Furthermore, the influence of roof shape and configuration of street canyon on characteristics of flow and pollutant distribution is also systematically studied, with the results shown in scenario B.

The main conclusions of the thesis are that the uncertainty of numerical results derives from different aspects. Wind profile in the inlet profile generated by morphometric method brings major error to the simulation results. Current turbulence models cannot compromise the simulation results between flow field and pollutant distribution field. Ignored small-scale obstacles also need to be handled carefully. Numerical results revealed that flow and pollutant distribution inside street canyons are mainly dominated by the geometry of the street canyon itself. Medium-spaced surrounding buildings are also better able to transport pollutant out of the street canyon. Through systematic analysis, roof shape is proven to have a significant effect on flow and pollutant distribution inside a street canyon. The major impact is altered turbulence intensity depth and strength of shear layer inside the street canyon, which is important for pollutant removal process out of the street canyon.

In the future, advanced turbulence models accompanied by small-obstacle effect models need to be developed in order to reliably simulate flow and pollutant dispersion simultaneously. Based on the advanced turbulence model, simulation of flow and pollutant dispersion in a complex 3D environment is essential in the next steps for the purpose of engineering application. Accurate vertical wind profile provided for inlet profile is another interesting direction for further development.
Keywords: Flow; Pollutant dispersion; CFD; Street canyon; Reliability
Preface

The work for this thesis was carried out between September 2009 and September 2012 at the Division of Building Service and Energy Systems in the Department of Civil and Architectural Engineering at the Royal Institute of Technology (KTH), Stockholm, Sweden. Special thanks are owed to the Chinese Scholarship Council (CSC) for funding the research.

My supervisor, Professor Ivo Martinac, deserves particular thanks for giving me the opportunity to conduct the research, guiding me into the field, and for his excellent encouragement. His questions, comments, and broad knowledge supported me throughout my research process. Professor Martinac also provided me with a large degree of freedom for my research and work, which made me become very independent in my thinking, producing, and organizing myself. I believe this experience will benefit me for the rest of my life.

I would also like to sincerely thank my co-supervisor, Professor Mats Sandberg, for his good suggestions during the research process. To my colleagues, Hai Lu, Juan Ren, Yihe Shen, and Qian Wang, thanks for the creative, harmonious and friendly working environment.

In addition, I want to gratefully acknowledge all of my friends for being with me through some fun and unforgettable times.

Finally I want to express my most heartfelt gratitude to my parents and my family for all their supports and encouragement during my overseas study. Your efforts are beyond any words.
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$x_i, x, y, z$</td>
<td>Coordinate axes in horizontal, lateral and vertical direction</td>
</tr>
<tr>
<td>$u_i, u, v$ and $w$</td>
<td>Velocity in horizontal, lateral and vertical direction</td>
</tr>
<tr>
<td>$U_i$</td>
<td>Time averaged of $i^{th}$ velocity component</td>
</tr>
<tr>
<td>$\bar{u}_i$</td>
<td>The $i^{th}$ fluctuating velocity component</td>
</tr>
<tr>
<td>$\bar{u}_d$</td>
<td>Diffusion velocity inside street canyon</td>
</tr>
<tr>
<td>$\bar{u}_c$</td>
<td>Advective velocity inside street canyon</td>
</tr>
<tr>
<td>$\bar{u}_t$</td>
<td>Turbulent diffusion velocity at roof level</td>
</tr>
<tr>
<td>$U(z)$</td>
<td>Mean velocity at vertical height</td>
</tr>
<tr>
<td>$\sigma_i$</td>
<td>Standard deviation of the $i^{th}$ velocity component</td>
</tr>
<tr>
<td>$p$</td>
<td>Hydrostatic pressure</td>
</tr>
<tr>
<td>$t$</td>
<td>Time coordinate</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density of air</td>
</tr>
<tr>
<td>$L_{ref}$</td>
<td>Reference length scale</td>
</tr>
<tr>
<td>$U_{ref}$</td>
<td>Reference velocity scale</td>
</tr>
<tr>
<td>$U_i^*$</td>
<td>Non-dimensional velocity</td>
</tr>
<tr>
<td>$x_i^*$</td>
<td>Non-dimensional coordinate</td>
</tr>
<tr>
<td>$\partial / \partial x_i$</td>
<td>Spatial partial derivative</td>
</tr>
<tr>
<td>$t^*$</td>
<td>Non-dimensional time</td>
</tr>
<tr>
<td>$p^*$</td>
<td>Non-dimensional pressure</td>
</tr>
<tr>
<td>$Re_{ref}$</td>
<td>Reference Reynolds number</td>
</tr>
<tr>
<td>$Pr_r$</td>
<td>Roof Prantle number</td>
</tr>
<tr>
<td>$Re_r$</td>
<td>Roof Reynolds number</td>
</tr>
<tr>
<td>$Pe_r$</td>
<td>Roof Pefect number</td>
</tr>
<tr>
<td>$\bar{u_i}$</td>
<td>Time-averaged fluctuate velocity</td>
</tr>
<tr>
<td>$\tau_{i,j}$</td>
<td>Shear stress</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Shear stress</td>
</tr>
<tr>
<td>$S_{i,j}$</td>
<td>Time-averaged rate of strain tensor</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulence kinetic energy</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Dissipation rate of turbulence kinetic energy</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Turbulence viscosity of air</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Viscosity of air</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity of air</td>
</tr>
<tr>
<td>$S_{ct}$</td>
<td>Turbulent Schmidt number</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Specific dissipation rate of turbulence kinetic energy</td>
</tr>
<tr>
<td>$l_u$</td>
<td>Turbulence intensity at horizontal direction</td>
</tr>
<tr>
<td>$z_0$</td>
<td>Roughness length</td>
</tr>
<tr>
<td>$d$</td>
<td>Zero-displacement height</td>
</tr>
<tr>
<td>$H, H_1, H_2$</td>
<td>Reference building height, building height for targeted street canyon</td>
</tr>
<tr>
<td>$W$</td>
<td>Width of street canyon</td>
</tr>
<tr>
<td>$L$</td>
<td>Length of street canyon</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Atmospheric boundary layer height</td>
</tr>
<tr>
<td>$z^*$</td>
<td>Rough wall region height</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Von Karman constant, set to 0.41</td>
</tr>
<tr>
<td>$u_{max}$</td>
<td>Maximum friction velocity in vicinity of building roof level</td>
</tr>
<tr>
<td>$u^*$</td>
<td>Friction velocity in inertial region</td>
</tr>
</tbody>
</table>
$u_{st}$  Local friction velocity
$k_s$  Roughness height
$k_s^+$  Non-dimensionial roughness height
$C$  Pollutant concentration
$K$  Non-dimensional pollutant concentration
$\lambda_F$  Frontal area index; i.e, flow facing roughness area per unit horizontal area
$\lambda_P$  Plan area index; i.e, horizontal built-up area per unit horizontal area

2D  Two-dimensionnal
3D  Three-dimensionnal
IRF  Isolated roughness flow
WIF  Wake interference flow
SF  Skimming flow
AR  Aspect ration $H/W$
Rb  Richardson number
$Q_e$  Source strength
# Table of contents

1 Introduction.................................................................................................................1  
   1.1 Motivation of the work..........................................................................................1  
   1.2 Objective and research questions........................................................................2  
   1.3 The methodology and model structure..............................................................3  
   1.4 Structure of the thesis........................................................................................3  
2 Numerical modeling: theory and discussion..............................................................5  
   2.1 Steady Reynolds Averaged Navier-Stokes Equations (RANS) and turbulence modeling.......................................................................................................................5  
   2.2 Boundary problems.............................................................................................8  
   2.3 General settings for numerical simulation.........................................................11  
   2.4 Model discussion..............................................................................................12  
   2.5 Reynolds number independence..........................................................................13  
   2.6 Results comparison between numerical simulation and experimental............14  
   2.7 Experiences of real urban case studies...............................................................15  
3 Rough-wall boundary layer flow ..............................................................................16  
   3.1 Classification of rough-wall boundary layer flow...............................................16  
   3.2 Mean flow structure..........................................................................................17  
   3.3 Turbulence structure........................................................................................21  
4 Literature review on wind flow and pollutant dispersion inside street canyons........24  
   4.1 Definition of idealized street canyons.................................................................25  
   4.2 Flow characteristics of inside idealized street canyons.......................................25  
   4.3 Characteristics of pollutant dispersion inside idealized street canyons................28  
5 Summary of research and discussion.......................................................................33  
   5.1 Scenario study A...............................................................................................33  
      5.1.1 Methodology..............................................................................................33  
      5.1.2 Physical model and oncoming wind profile analysis....................................34  
      5.1.3 Numerical model validation.......................................................................36  
      5.1.4 Numerical simulation results and discussions............................................45  
      5.1.5 Conclusions and discussion......................................................................48  
   5.2 Scenario study B...............................................................................................49  
      5.2.1 Methodology..............................................................................................50  
      5.2.2 Simulation results and discussion..............................................................53  
      5.2.3 Conclusions and discussion......................................................................58  
6 Conclusions and future work....................................................................................60  
7 References..............................................................................................................64
Chapter 1 Introduction

1.1 Motivation of the work

An atmospheric boundary layer is defined as a layer directly influenced by surface friction. It is generally classified into three sub-layers: roughness, inertial, and mixed (Figure 1.1). The turbulence in the roughness sub-layer is fully three-dimensional and significantly depends on the properties of roughness at the surface. The oncoming wind flow is retarded by the ground buildings, parks, and trees. Wind flow inside the roughness sub-layer then becomes three-dimensional and turbulent. In various climate conditions, the inertial sub-layer will compress the roughness layer. As a result, the roughness boundary could also be called an urban canopy layer. The height-scale of the urban canopy layer is the approximately building height. The flow in the inertial sub-layer mainly depends on the degree of open roughness surface retarding the wind flow. The flow field and pollutant dispersion inside layer will directly influenced by atmospheric stability, which is normally ruled by Monin-Obukhov similarity theory. The turbulence generated by buoyancy effect dominates the mixed layer, while wind-induced shear stress dominates the urban canopy layer and inertial sub-layer (Rotach, 1993; Easong, 2000; Roth, 2000; Bernard et al., 2005).

![Figure 1.1 Atmospheric boundary layer structure in urban area](image-url)

There is a worldwide trend toward urbanization, accompanied by serious social and environment problems, including severe air pollution. This is particularly in developing countries such as China, and India. As concerns increases about the health of urban occupants, microclimate and urban air quality has gained attention in recent years. While governments increasingly control large industrial sources of pollutants, motor traffic is rapidly increasing. Traffic pollutants, such as nitrogen dioxide (NO$_x$), sulphur dioxide (SO$_2$), carbon monoxide (CO), and airborne lead (Pb), are major emission sources in urban areas (Fenger, 1992). Airborne lead is the most harmful particulate matter for children. It can results in permanent brain damage and other diseases for residents exposed to a hazardous environment. Therefore, traffic-induced pollutant dispersion have attracted particular interest in urban area, particularly pollutant distribution around building envelops and streets.

Vehicle pollutant emissions are usually inside the canopy layer, where the atmospheric layer has serious interference from buildings and other roughness elements, such as cars, and gardens. Characteristics and transportation of pollutants changes significantly depending on surrounding density, height and shape of buildings, or roughness elements. In real urban circumstances, flow inside and above the urban canopy layer varies due to complex geometry
structure and local weather, therefore properties of flow inside and above canopy layer also demonstrated unsteady. As a consequence, flow field and wind-induced pollutant dispersion in the roughness sublayer for neutral stratified atmospheric conditions are constrained in the present research topic.

Investigation of flow and pollutant dispersion has been widely documented in the literature. Pollutant dispersion modeling, which quantifies and predicts air pollution levels, as well as temporal and spatial variations, can be classified into three types: operational models, full-scale tests or wind-tunnel experiments, and numerical simulations (Vardoulakis, et al., 2003). Operational models are developed from the systematic parameters and wind tunnel experimental databases, such as Gaussian plume models, CALINE4, TNO and CAR, STREET-SRI, CPBM, OSPM and AEOLIUS, and Reception model (Berkowicz et al., 1997). These operational models are easy to use and provide fast assessment of air quality in urban environments. The empirical constants in the model are widely based on measurements and wind-tunnel experiments. However, these models lack detailed information about flow patterns in the canyon layer, which means that they are mainly used as regulation models. Better prediction of pollutant dispersion and air quality levels highly depends on the database and a good understanding of pollutant characteristics.

Full-scale tests and wind-tunnel experiments are preferable methods to evaluate air quality. However they have high costs they must repeat typical circumstances in reality, as well as equipment costs for measurements. As a result, they are mainly used as validation tools.

Numerical modeling generally is a term for the computational fluid dynamics (CFD) involved in a system analysis of fluid flow, heat transfer, and associated phenomena (such as pollutant transfer processes). CFD simulation is a micro- and cost-effective tool to assess air pollutant dispersion and flow patterns through visible demonstration. However, it still faces huge challenges, such as modeling and realistic boundary settings. On the one hand, CFD can achieve more visible, detailed flow, and pollutant distribution around buildings with increasing computational power. On the other hand, CFD simulation domains cannot cover an entire city. Therefore, an accurate and reliable building computational domain is also a challenge for wind engineering.

1.2 Objective and research questions

The current project aims to establish reliable mathematical models of air quality prediction and investigate the impact of building geometry and configuration on airflow patterns and pollutant concentrations in street canyons. For the purpose of engineering application, reliable wind profile data as input for wind-induced issues will also be an important task in this research. This research will provide architects and engineers with guidelines for planning and designing urban building shapes from perspective of air quality. The work can also be the basis of further research on complex three-dimensional wind-induced flow and pollutant dispersion issues in the atmospheric boundary layer (ABL).

The work addresses the following research questions:
What are the airflow characteristics and traffic-induced pollutant characteristics in the urban street impacted by local building geometry and surroundings? And what are the issues for numerical research in ABL?

The following parameters are also considered in the mathematical modeling:

- Location of buildings
- Outdoor environment conditions
- Density of built-up areas
- Street canyon parameters (Aspect ratio H/W and length to width ratio L/H)
- Roof shapes

The main question is broken into the following sub-questions:

- What is the state-of-art of knowledge of flow patterns and pollutant dispersion characteristics inside street canyons?
- Does density of built-up area influence flow field and pollutant dispersion characteristics inside objected street canyon?
- How does roof shape impact flow field and pollutant dispersion inside street canyons?
- What are the weaknesses of mathematical modeling of air quality prediction?

1.3 The methodology and model structure

Figure 1.2 summarizes the methodology. The thesis deals with air quality problems in street canyons caused by pollutants from vehicle emissions, without considering photo-chemical reaction. The thesis combines the micro-mathematical CFD and empirical models (meteorology). Application of the model is used to analyze the effects of upcoming wind flow and geometry of street canyons on flow characteristics and pollutant concentration inside two-dimensional street canyons (Scenario A). Since upcoming wind flow is strongly altered by local roughness elements, this thesis also investigates the influence of roof shapes on flow pattern and pollutant dispersion (Scenario B).

![Figure 1.2 Methodology and modeling of research](image)

1.4 Structure of thesis
Chapter 2 presents the entire numerical model, including the turbulence model, wall function theory, numerical method, and convergent criteria. This section also presents drawbacks of turbulence models between RANS and LES. Chapter 3 discusses the fundamentals of the research boundary, field-test and wind-tunnel experimental data on mean velocity profiles, turbulence properties above the roughness surface on rural and urban areas. Chapter 4 reviews the state-of-art knowledge of flow pattern and pollutant dispersion in street canyons. Chapter 5 presents results and summaries. This section also discusses future research and the limitations of current research. Finally, Chapter 6 answers the proposed research questions.
Chapter 2 Numerical Modeling: Theory and Discussion

2.1 Steady Reynolds-Averaged Navier-Stokes Equations (RANS) and turbulence modeling

Since the complexity of flow patterns and pollutant dispersion characteristics impacted by buildings, numerical model is introduced to analyze flow and scalar-transportation phenomenon in surface sub-layer. The coordinate system of the numerical model is defined as follows. The positive longitudinal direction is given by flow direction. The vertical direction is given upward from the surface, originating at buildings’ ground floor of buildings. Lateral direction is given as a right-hand orthogonal coordinate system. The wind velocities in the coordinate system at longitudinal, lateral, and vertical directions are labeled as \( u, v \) and \( w \), respectively. In order to easily represent novel velocity quantities, \( u_i = U_i + u'_i \) is given in the model, and \( x_i \) represents the coordinate axis in different directions. Time-averaged mean velocities are referred to as \( U_i \), and instantaneous turbulent quantities are represented as \( u'_i \).

The term fluid flow problems refer to the solution of series of differential equations of Navier-Stokes and continuity equations, with appropriate boundary conditions. These equations are derived from Newton’s Second Law and describe with the conservation of momentum, combined with the continuity equation, which are used to solve flow variables issues. The typical Navier-Stokes equation is expressed as:

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

In the model, body force, such as Coriolis force, is ignored because they are very small at ground level. Other body forces, such as gravity force, are also ignored in the current research. Non-dimensional Navier-Stokes equations are introduced to give insight into all terms in the equation. Based on reference length scale \( L_{\text{ref}} \) and reference velocity scale \( U_{\text{ref}} \), the time scale is referred to as \( L_{\text{ref}} / U_{\text{ref}} \). Pressure term scale is referred to as \( \rho U_{\text{ref}}^2 \). All dimensional variables transform into non-dimensional form as:

\[
U_i^* = u_i / U_{\text{ref}}, x_j^* = x_j / L_{\text{ref}}, t^* = t / (L_{\text{ref}} / U_{\text{ref}}), p^* = p / (\rho U_{\text{ref}}^2)
\]

Therefore, the non-dimensional Navier-Stokes equation is

\[
\frac{\partial u_i^*}{\partial t^*} + u_j^* \frac{\partial u_i^*}{\partial x_j^*} = - \frac{1}{\rho} \frac{\partial p^*}{\partial x_i^*} + \frac{1}{Re_{\text{ref}}} \frac{\partial}{\partial x_j^*} \left( \frac{\partial u_i^*}{\partial x_j^*} + \frac{\partial u_j^*}{\partial x_i^*} \right)
\]

Where \( Re_{\text{ref}} \) defined as

\[
Re_{\text{ref}} = \frac{\rho L_{\text{ref}} U_{\text{ref}}}{\mu}
\]

As seen in Equation 2.3, background flow field is dominated by \( Re_{\text{ref}} \). Only at the same Reynolds number \( Re_{\text{ref}} \), can flow fields be similar to each other. This allows the experiments to be conducted at a smaller scale, based on the assumption that excluded physical effect in non-dimensional equations are not important for the flow field. However, this presents a problem particularly for reproducing atmospheric boundary layer in wind-tunnels. At smaller scales of prototype, higher velocity is needed to ensure Reynolds number independent.
In a real environment, the wind field is a time-variable. It is impossible to directly solve wind field. In order to solve the equations, Reynolds averaged method is introduced and an assumption is made that time-averaged turbulent velocity quantities \( \tilde{u}_i \) are equal to zero

\[ \tilde{u}_i = 0 \]  
(2.5)

Therefore, the general form of the three-dimensional, incompressible Reynolds-averaged Navier-Stokes (RANS) Equation is transformed as follows:

\[ \frac{\partial \tilde{u}_i}{\partial t} + u_j \frac{\partial \tilde{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j}\left(\mu \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) - \tilde{u}_i \tilde{u}_j\right) \]  
(2.6)

The continuity equation is also transformed into time-averaged form as

\[ \frac{\partial \tilde{u}_i}{\partial x_i} = 0 \]  
(2.7)

In order to solve RANS, closure equations are proposed attempted to solve Reynolds stress terms \(-\tilde{u}_i \tilde{u}_j\). Based on the assumptions of Boussinesq approximation and isotropic eddy viscosity, Reynolds stress is stated linearly to the rate of fluid deformation with coefficient \( \mu_t \)

\[ \tau_{ij} = -\rho \tilde{u}_i \tilde{u}_j = \mu_t S_{ij} \quad (S_{ij} = \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}) \]  
(2.8)

The term \( S_{ij} \) represents the time-averaged rate of strain tensor. For the purpose of estimating the value of turbulence viscosity \( \mu_t \), a series of turbulence models are proposed. The widely used two-equation eddy viscosity turbulence model, called standard \( k-\epsilon \) (Lauder and Spalding, 1974), solved the following transport equation of turbulent kinetic energy (TKE) \( k = \frac{1}{2} \tilde{u}_i \tilde{u}_i \). We integrated another variable, called turbulence dissipation rate \( \epsilon = 2 \nu S_{ij} S_{ij} \) into model

\[ \frac{Dk}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j}\left(\mu + \mu_t / \sigma_k\right) \frac{\partial k}{\partial x_j} - \tilde{u}_i \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} - \epsilon \]  
(2.9)

\[ \frac{D\epsilon}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j}\left(\mu + \mu_t / \sigma_\epsilon\right) \frac{\partial \epsilon}{\partial x_j} - c_{\epsilon 1} \frac{\epsilon}{k} \tilde{u}_i \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} - c_{\epsilon 2} \frac{\epsilon^2}{k} \]  
(2.10)

The turbulence viscosity \( \mu_t \) is then modeled as

\[ \mu_t = \rho C_{\mu f} \mu \frac{\epsilon^2}{k} \]  
(2.11)

Typical constants in the equations are listed in Table 2.1. Typical values of constants \( C_\mu, \sigma_k, \sigma_\epsilon, c_{\epsilon 1} \) and \( c_{\epsilon 2} \) in the equation are 0.09, 1.0, 1.3, 1.44 and 1.92, respectively. The model is widely used for various industrial applications. However the standard \( k-\epsilon \) turbulence model is still deficient in that it is imprecise in accounting for viscous effect at near wall region (Bredberg, 2001).

The turbulence of scalar transport must also be modeled. Since vehicle-emitted pollution is investigated in urban atmospheric boundary layer, which is perceived as gas, species transport equation is modeled as
\[
U_j \frac{\partial c_a}{\partial x_j} = \frac{\partial}{\partial x_j} \left( D_a - u_j'c_a' \right) \frac{\partial c_a}{\partial x_j} + S_a \tag{2.12}
\]

\[
\frac{\partial}{\partial x_j} \left( u_j'c_a' \right) = \nu c \frac{\partial c_a}{\partial x_j} \tag{2.13}
\]

\[
-\bar{u}_i'\bar{u}'_j = \nu c \frac{\partial u_j}{\partial x_j} - \frac{2}{3} k \delta_{ij} \tag{2.14}
\]

\[
S_{C_t} = \frac{\nu c}{\nu c} \tag{2.15}
\]

The term \( x_j \) is Cartesian coordinate, \( u_j \), and \( u_j' \) represent mean velocity and fluctuating velocity. The term \( p \) is mean pressure, \( \rho \) is density of flow, \( \mu \) is flow viscosity. The terms \( k, \varepsilon \) and \( P_k \) respectively represent turbulence kinetic energy, dissipation rate and kinetic energy production. The terms \( C_a, C_a' \) and \( S_a \) respectively represent species mean concentration, fluctuate concentration and source strength. \( \nu_t \) and \( \nu_c \) represent turbulence viscosities of momentum and pollutant concentration, separately. The terms \( S_{C_t}, D_a \) and \( \delta_{ij} \) respectively represent turbulent Schmidt number, species molecular diffusivity and Kronecker delta.

Typical values of the turbulence model for wind flow and pollutant dispersion study are listed as Table 2.1.

| Table 2.1 Turbulence model constant value (Lauder and Spalding, 1974) |
|----------------|---|---|---|---|---|
| Constant | \( C_\mu \) | \( \sigma_k \) | \( \sigma_\varepsilon \) | \( c_{\omega 1} \) | \( c_{\omega 2} \) | \( S_{C_t} \) |
| Value | 0.09 | 1.0 | 1.3 | 1.44 | 1.92 | 0.7 |

Further developed turbulence models, such as Renormalisation Group (RNG) \( k - \varepsilon \) (Yakhot and Orszag, 1986, 1992), Realizable \( k - \varepsilon \), Low Reynolds Number \( k - \omega \) (Wilcox, 1988), Shear Stress Transport (SST) \( k - \omega \), are (all based on RANS) attempting to solve Reynolds Stress Terms \( -\bar{u}'_i\bar{u}'_j \) with assumptions of Boussinesq approximation and isotropic eddy viscosity (Fluent User Guide, 2005). From a mathematical standpoint, RNG \( k - \varepsilon \) and Realizable \( k - \varepsilon \) only differ from standard \( k - \varepsilon \) model by their constant values.

Unlike eddy viscosity expressed as product of a velocity and length scale in series of \( k - \varepsilon \) turbulence models, the \( k - \omega \) turbulence model refers to a reciprocal of time scale and a velocity scale to represent eddy-viscosity (Bredberg, 2001). This secondary quantity \( \omega \) is also more commonly perceived as specific dissipation rate of turbulence kinetic energy \( (\omega = k/\varepsilon) \). The advantage of the turbulence \( k - \omega \) model is that it can provide reasonable results without a wall function at the near wall region. In Fluent 14.0, a damping factor \( \beta^* \) is given to represent \( \omega = k/\beta^* \varepsilon \) and turbulent viscosity is modeled as \( \nu_t = \alpha^* \frac{k}{\omega} \). The transport equations of \( k - \omega \) are modeled as:

\[
\frac{Dk}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \mu + \mu_t/\sigma_k \right) \frac{\partial k}{\partial x_j} - \bar{u}_i'\bar{u}'_j \frac{\partial u_i}{\partial x_j} - \frac{k}{\beta^* \omega} \tag{2.16}
\]

\[
\frac{D\omega}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \mu + \mu_t/\sigma_\omega \right) \frac{\partial \omega}{\partial x_j} - c_{\omega 1} \frac{\omega}{k} \bar{u}_i'\bar{u}'_j \frac{\partial u_i}{\partial x_j} - c_{\omega 2} \omega^2 \tag{2.17}
\]

Typical constant values of turbulence model are

| Table 2.2 Turbulence model constant values |
|----------------|---|---|---|---|---|
| Constant | \( \alpha^* \) | \( \beta^* \) | \( \sigma_\omega \) | \( c_{\omega 1} \) | \( c_{\omega 2} \) | \( \sigma_k \) |
| Value | 1.0 | 0.09 | 1.5 | 0.52 | 0.072 | 1.0 |
As stated by Menter (1994), sensitivity of the original $k - \omega$ turbulence model to free stream $\omega_f$ is significant and may cause severe discrepancy in complex flow. Therefore, Menter (1994) developed a more advanced model named shear stress transport (SST) $k - \omega$ turbulence model. A transport equation of shear stress is introduced to improve results of adverse pressure gradient flow. By mathematical transformation, the SST $k - \omega$ turbulence model with blending functions can solve the flow at near wall region as $k - \omega$ turbulence model. It can also be treated as a transformed $k - \varepsilon$ turbulence model at the far-wall region (Fluent user guide, 2005). Simulation results of SST model is independent of $\omega_f$. As the advantages of considering the viscous effect at the near wall region and the easy of integration with other turbulence models become clearer, $k - \omega$ turbulence models are concerned with developing of more advanced turbulence models.

Reynolds stress turbulence model (RSM) is another type of turbulence model. Unlike eddy viscosity turbulence models that use isotropic eddy viscosity hypothesis to closure the transport equation, RSM directly models all Reynolds stress terms separately to consider anisotropic effect (Fluent user guide, 2005). However they are also challenges for the simulation.

LES generally provides better results than RANS models (Tominaga, 2007, 2010, 2011). However, the LES model spends substantially more running time than do RANS models, especially for big urban areas. Furthermore, LES models require time- and space-wind data as boundary conditions to properly simulate the flow. However, this data is rarely available in practice (Franke et al., 2007). Since there are challenges with these two series of turbulence models, they are not the focus of this thesis.

2.2 Boundary problems

Boundary conditions, including inlet profile, wall boundaries, upper boundaries and outlets, are important factors for proper simulation of atmospheric boundary flow. For wall boundaries, most turbulence models are developed for fully turbulent flow. However, when the computational region comes to the near wall area, the viscosity effect will increase influence on flow field over turbulence viscosity. Two methods are available to consider this situation: the wall function method and the low-Reynolds number turbulence model. If low-Reynolds is used at the near wall region, a high-density grid should be generated to resolve larger gradients of energy dissipation. Therefore this model requires substantial computational resources and time. The wall function method then can save computational time and get acceptable simulation results (Bredberg, 2001; Fluent User Guide, 2005). The section will focus on this method. Researches on properly setting boundary conditions for the simulation will also be discussed in this section.

Inlet profile

In a steady, incompressible two-dimensional flow field, the vertical wind profile is described as logarithm law. Measurements of turbulent kinetic energy budgets revealed that production of kinetic energy in the logarithm region equal rate of dissipation. Based on the assumptions of wind profile, turbulent kinetic energy and dissipation rate in the logarithm region are expressed as (Richards and Hoxey, 1993; Bredberg, 2001):
\[ U(z) = \frac{u^*}{\kappa} \ln \left( \frac{z+z_0}{z_0} \right) \]  
(2.18)

\[ k = \frac{u^*}{\sqrt{c^\mu}} \]  
(2.19)

\[ \varepsilon = \frac{u^*}{\nu(z+z_0)} \]  
(2.20)

We only need to satisfy the following constrained condition

\[ \sigma_\varepsilon = \frac{K^2}{(C_{e2}-C_{e1})/\sqrt{c^\mu}} \]  
(2.21)

The inlet profile of constant, turbulent, kinetic energy production with height that was proposed by Richards and Hoxey (1993) has been widely used in the computational wind engineering. Recently, Yang et al. (2009) proposed a variable kinetic energy profile with height and a corresponding dissipation rate profile. However, it did not consider the momentum equation. Furthermore, Yang’s solution still strongly relies on experimental results, so it is not advantageous for industrial applications.

**Wall boundary condition**

Wall function is normally based on a universal assumption that constant shear stress exists and that the length scale of a typical eddy is proportional to the distance from the wall. Thus, the flow field yields

\[ (\mu + \mu_t) \frac{\partial U}{\partial z} = \rho u^* \]  
(2.22)

Variable \( \mu \) is fluid dynamic viscosity, \( \mu_t \) is eddy viscosity, \( U \) is mean velocity, \( u^* \) is friction velocity, and \( \rho \) is fluid density. This equation also applies to the wall surface to maintain the flow equilibrium. Wall surface shear stress is equal to \( \tau_{wall} = \rho u^* \). Equation (2.22) transfers into dimensionless form as:

\[ (1 + v^+ \mu_t) \frac{dU^+}{dz^+} = 1 \]  
(2.23)

The dimensionless form is defined by \( U^+ = \frac{U}{u^*}, z^+ = \frac{u^* y}{v} \) and \( v^+ = \frac{v}{v} \). In general, near wall flow fields can be divided into three parts: linear layer, buffer field and logarithmic layer. In the linear layer, the viscosity force is much greater than turbulent kinetic viscosity \( (v = \frac{\mu}{\rho}) v \gg \nu_t \), so the approximation relation of velocity distribution is \( U^+ = z^+ \). In the logarithmic layer, turbulence viscosity is much greater than viscosity \( v \ll \nu_t \), so the velocity distribution is

\[ v^+ \frac{dU^+}{dz^+} = 1 \]  
(2.24)

Integrated as logarithmic law:

\[ U^+ = \frac{1}{\kappa} \ln(z^+) + B \]  
(2.25)

The linear law or laminar law is valid for \( y^+ < 5 \). The logarithmic law \( y^+ \) is valid above 30 up to 500 to -1000. Constant \( B \) is in the range of 5.0~5.2. This means the first grid non-dimensional height should stay in the logarithmic area when using the wall function. Nikuradse presented a modified surface function based on experimental data on various
roughness surfaces. A new parameter, called *dimensionless sand-grain roughness height* 
\[ k_s^+ = \frac{u_* k_s}{v} \], is introduced (Blocken et al., 2007). The modified function is expressed as

\[ U^+ = \frac{1}{k} \ln(z^+) + B - \Delta B(k_s^+) \quad (2.26) \]

Most commercial CFD software only affords \( k_s \) wall function. The wall function in Fluent 6.2 is given by:

\[ \frac{U_p u_*}{u_T^2} = \frac{1}{k} \ln\left( \frac{E u_* z_p}{u_T^2 (1 + C_S k_s^+)} \right) \quad (2.27) \]

The wall function distinguished by the flow shear stress \( u_* \), and wall shear stress \( \tau_{wall} = \rho u_*^2 \). \((1 + C_S k_s^+)\) is used to modify surface roughness. Constant \( C_S \) is a modification coefficient, ranging between 0 and 1. For different surface roughnesses, the different \( k_s^+ \) values can be chosen. Since dynamic parameter roughness length \( z_0 \) is used in inlet profile, the following relationship between equivalent sand-grain height \( k_s \) and corresponding dynamic roughness length \( z_0 \) needs to fulfilled to match with inlet profile and wall function (Blocken et al., 2007)

\[ k_{s,ABL} = 30z_0 \quad (Theory) \quad (2.28) \]
\[ k_{s,ABL} = 9.783z_0/C_S \quad (Fluent) \quad (2.29) \]

However, this requirement conflicts with another requirement. At the ground level, a high solution mesh should be generated with the following principle: The distance of the center point of the wall-adjacent cell to the wall (bottom of domain) ≥ the physical roughness height \( k_s \) of the terrain. For instance, a dynamic roughness height 1.0 is given which is suggested by Wieringa (1992) for a suburban area of a city. Then, equivalent sand-grain roughness height \( k_s \) should between 10 (Fluent with \( C_S = 1.0 \)) and 30 (CFX). Obviously, it is impossible to set the first cell height to such a great value in order to fulfill the high-solution mesh requirements. Therefore, if improper first grid height is set in the grid meshing, it would cause untended deceleration or acceleration approaching flow (Blocken et al., 2007).

Based on this analysis, Blocken et al. (2007) concluded that the wall function is based on the smooth wall theory. However in an urban environment, the ground roughness length is too big, so smooth wall theory is invalid. At same time, Blocken et al. (2007) also proposed several remedial approaches to eliminate this inconsistency between inlet-flow profile and wall function. Thise remedial approaches include varying the first cell height according to the ground environment (Franke et al., 2004), explicitly modeling the roughness element (Miles and Westbury, 2003), minimizing the approach solution domain (Blocken et al., 2006), generating an inlet profile of ABL flow by the standard \( k_s \) wall function (Blocken et al., 2004), verifying inlet turbulence kinetic energy (Blocken and Carmeliet, 2006) or forcing a consistent wall shear stress with ABL inlet profile at bottom of upstream flow.

**Top boundary condition**

Most engineering applications use free slip condition at the top field. Hargreaves and Wright (2007) pointed out the reasons for the decay of mean velocity and turbulent kinetic energy by fetching for the approach flow. Not only the modified wall function, but also the top boundary conditions resulted in this situation. Hargreaves and Wright noted that the top boundary of the solution domain should have a constant shear stress \( \tau = \rho u_*^2 \) in order to
develop equilibrium ABL. Richards and Hoxey (1993) first noted this, but it has been ignored by many researches. Based on the advanced work of Yang et al. (2009) and Hargreaves and Wright (2007), Sullivan et al. (2011) proposed a new top boundary condition combined with Yang’s work that turbulent kinetic energy production varies with height at top of boundary. A constant shear stress exists that allows the outfield flow in and out of solution domain.

However this method is also still loosely based on experimental data, which is hardly applicable in real urban cities. An important concept proposed by Sullivan et al. (2011) is that the proper boundary condition specified on the top field is critical point to generate equilibrium ABL. An alternative is applying wind velocity, turbulent kinetic energy and dissipation rate directly derived from the inlet profile. Blocken et al. (2007) used this method.

Outlet boundary condition

It is normally assumed that flow at the outlet boundary is fully developed. Therefore, the diffusion fluxes for all flow variables are zero in the exiting direction. Normal gradients of variable are also zero at outlet boundaries (Fluent 6.2 User Guide, 2005). This is expressed as:

$$\frac{\partial}{\partial x} (u, v, w, k, \varepsilon) = 0$$

(2.30)

2.3 General settings on choice of computational domain, grid, turbulence models and boundary conditions for flow field and pollutant dispersion

As an objective for COST 732 (Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment, COST 732, 2007) to improve and assure quality of micro-scale meteorological models that predict flow and pollutant dispersion in urban or industrialized areas, an exploratory analysis was conducted. Excel-based analysis tools and best guidelines for CFD simulation are released.

In the guidelines, the assumed objected building height is H, and the length of upstream fetch should stay between 6-10H to confirm the region of interest. The computational domain has no interaction with the inlet profile, while errors introduced by mismatch between wall function and inlet profile are minimized. For the extension of domain in the flow direction, a distance of 5H or even longer should be given. The lateral length should also give at least 5H to avoid blockage effect. Because the flow will not be affected by ground obstacles more than distance 5H in the vertical direction, a distance of 5H should be set.

The grid, used to discretize the computational domain, is important in reducing errors and achieving precise results for the Finite Volume Method and Finite Differential Method in RANS turbulence models. The resolution of the grid should be fine enough to capture the important physical phenomena such as shear layer and vortices. The expansion ratio between two consecutive cells should be below 1.2 in order to reduce truncation error introduced by grid stretching/compression in high-gradient regions. High-order numerical schemes can also decrease the error. At this point, a hexahedral grid is preferable to a tetrahedral, as the latter introduces small truncation errors.

General settings for applied CFD simulation to flow field and pollutant dispersion are described as follows: Among steady-state RANS turbulence models, the standard $k - \varepsilon$ model is the most common. The standard wall function is matched with the standard $k - \varepsilon$ model for
predicting pollutant dispersion in street canyon. Theodoridis and Moussiopoulos (2000) discovered that two-zonal model has better ability than standard $k - \varepsilon$ model to predict pollutant concentration in street canyons. The latter overestimates maximum pollutant concentration. In the paper, a series of $k - \omega$ turbulence models will be used for comparison.

At the inlet surface, logarithm law and equilibrium profile are more frequently used by researchers if there is no wind-tunnel measurement data available. While at outlet surface, the gradient of all parameters usually is usually set to zero. A symmetry condition is prescribed at top of domain only if the vertical domain was higher enough which will not influence interest area (Sullivan et al., 2011). From a physical viewpoint, alternatives are fixed values extracted from the inlet profile or constant shear stress prescribed at top of domain (Blocken et al., 2007). In three-dimensional cases, the zero gradient boundary condition or symmetry is defined at the lateral of domain. The governing equations are discretized by finite volume method. A SIMPLE algorithm is used for solve pressure-velocity coupling, second-order upwind theme for momentum and turbulence model equations. If unsteady simulation is applied, second-order method should also be chosen to approximate time derivations. Finally, convergence criteria of the scaled residuals for all variables and continuity equation are set as $10^{-4}$, except for the pollutant concentration $10^{-6}$ (Fluent User Guide, 2005).

<table>
<thead>
<tr>
<th>Items</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Software version</td>
<td>ANSYS Fluent 14.0</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Standard $k - \varepsilon$ model with constants $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, and $Sc_t = 0.7$</td>
</tr>
<tr>
<td>Inlet boundary</td>
<td>Proposed model stated in above section</td>
</tr>
<tr>
<td>Outlet boundary</td>
<td>Outflow $\frac{\partial}{\partial x} (u, v, w, k, \varepsilon) = 0$</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td></td>
</tr>
<tr>
<td>Upper boundary</td>
<td>Constant values of velocity, turbulent kinetic energy and turbulence dissipation rate extracted from inlet profile</td>
</tr>
<tr>
<td>Ground boundary</td>
<td>Standard wall function</td>
</tr>
<tr>
<td>Wall boundary</td>
<td>Standard wall function with defined wall roughness</td>
</tr>
<tr>
<td>Discretized method</td>
<td>Finite volume method</td>
</tr>
<tr>
<td>Numerical method</td>
<td>Second-order upwind for all differential equations</td>
</tr>
<tr>
<td>Convergent criteria</td>
<td>$\frac{</td>
</tr>
</tbody>
</table>

2.4 Model discussion

The $k - \varepsilon$ turbulence model has been widely calibrated and tested for the industrial flows, which involved flow separation cases. For atmospheric boundary flow, most constants have similar values to industrial flows. However constant $C_\mu$ exhibited a large difference since no agreed constant value in turbulence closure model was accepted in the industrial flow. This also has problems for atmospheric boundary flow simulation. The inlet turbulent kinetic energy ($k$) is important for the pollutant dispersion in the $k - \varepsilon$ turbulent model. The standard $k - \varepsilon$ model uses the high-frequency part of the power spectra, together with Kolmogorov’s theory, to derive turbulent kinetic energy ($k$). However there is no reason to assume that this approach is applicable within the roughness sublayer.
Richards and Hoxey (1993) illustrated that the cut-off high-frequency has a negligible effect on Reynolds stress $-u'w'$, but introduced errors to turbulent kinetic energy estimation. Hoxey (1993) proposed an inlet profile of mean velocity ($U(z) = \frac{u_*}{\kappa} \left( \ln \left( \frac{z+z_0}{z_0} \right) \right)$), turbulent kinetic energy ($k = \frac{u_*^2}{\sqrt{C_\mu}}$), and turbulent dissipation rate ($\varepsilon = \frac{u_*^3}{\kappa(z+z_0)}$) for neutral stratified conditions with the two-dimensional standard $k-\varepsilon$ model. This is based on the principle that high-frequency part of the power spectra together with Kolmogorov’s theory was used to derive turbulent kinetic energy ($k$). The value of constant $C_\mu$ was 0.09 along the vertical direction. Turbulent kinetic energy at inlet profile is then $k = 3.33u_*^2$.

However, as shown in Table 3.2, turbulent kinetic energy is then constrained into $k(z) = 5.5u_*^2$ for higher layer and $k(z) = 3.14u_*^2$ for the layer in the vicinity of building roof level. It introduces a mismatch problem between the inlet profile and the numerical model since the turbulence model normally sets turbulence constant $C_\mu$ as 0.09. This incompatibility between inlet profile and turbulence model is documented in the literature (Richards and Hoxey, 1993; Yang et al., 2009; Gorle et al., 2009, 2010; Parente et al., 2011).

Several methods have been proposed to improve the mismatch of inlet profile and turbulence model. The most investigated method directly revised turbulence constants in the turbulence model (Richards and Hoxey, 1993; Gorle et al., 2009, 2010). Yang et al. (2009) proposed a new inlet profile based on transport equation of turbulent kinetic energy without the considering momentum equation. Adding a source term into turbulent models (Parente et al., 2011), has also been proposed to use variable $C_\mu$ in the turbulence model. However, these solutions still face validity drawbacks, therefore preventing practical application of atmospheric boundary layer flow simulation. Nevertheless, improving turbulence modeling should be a concern for future research.

### 2.5 Reynolds number independence

In numerical scaling, Reynolds-number independence is required to reproduce realistic urban phenomena. Therefore, to approach flow similarity, the relationship between numerical models and full-scale simulations should hold to three principles: geometric scale, velocity scale, and Reynolds number scaling (ASCE Manuals and reports on Engineering Practice No.67, 1999). The geometric scale of building should fulfill the following:

$$\left( \frac{L_b}{z_0} \right)_m = \left( \frac{L_b}{z_0} \right)_p$$

(2.31)

in which $L_b$, and $z_0$ respectively represent a characteristic dimension of the building or structure, and the aerodynamic roughness of the terrain. The subscripts $m$ and $p$ respectively refer to model and prototype. The relationship between numerical scale and full-scale should equal the model and prototype ratios of overall building dimensions to the important meteorological lengths of the modeled approach wind.

However strict scaling of the mean wind, small geometry and turbulence Reynolds number for the approach flow is generally impossible for scaled numerical studies. The Reynolds number, independent of the approaching flow, can be assured by $\frac{u_*z_0}{\nu} > 2.5$. The geometric scale used in study ABL is typically about 1:100 or greater. The selection of
velocity scale is relatively arbitrary, as long as the model and full-scale flows remain aerodynamically similar, or independent of the building’s Reynolds number $Re_b$. Fortunately, distortion of the flow and the resulting variation in pressure distribution are usually considered negligible for Reynolds number $Re_b = V_bL_b/\nu$ in excess of 10000 (Hoydys et al., 1974; Meroney et al., 1996). Therefore, it is very important for Reynolds number to exceed the minimum number in the model. In this situation, viscous-induced effect can also be highly suppressed. However there is still a large gap between the full-scale Reynolds number and the numerical scale model. Validation process is generally needed by comparing numerical results and full-scale data. If validation data is originally from a wind-tunnel experiment, an equal size of the physical model is suggested to maintain dynamic similarity with wind-tunnel experimental settings.

2.6 Comparison of simulation results among turbulence models and experiments

Leitl and Meroney (1997) made a comparison between experimental results and numerical simulation results using the standard $k – \varepsilon$ turbulence model and the RNG $k – \varepsilon$ turbulence model. It revealed that all results well agreed for flat roofs but not slanted roofs. It proved that the universal constants of the turbulence model are not proper for the complex flow recirculation region. Discrepancies between experimental results and numerical turbulence models can achieve up to 90%. The standard $k – \varepsilon$ and RNG $k – \varepsilon$ models had no distinct difference for the flow simulation inside street canyons. It also showed that two-dimensional results of pollutant concentration inside street canyons are not as reliable as existing of lateral flow inside street canyons. Three-dimensional simulation should be introduced into future research.

Chan, et al. (2004) reported that the RNG $k – \varepsilon$ turbulence model had better agreement results than the standard $k – \varepsilon$ model or the realizable $k – \varepsilon$ model for isolated street canyons. Hanna et al. (2004) concluded that average pollutant concentration simulated by the standard $k – \varepsilon$ turbulence model was less than 36% of the observed experimental data, and the maximum value was even less than twice to observed data. Milliez and Carissimo (2007) also carried out a comparison between the standard $k – \varepsilon$ model and observed data. They discovered that standard $k – \varepsilon$ model overestimated turbulent energy for the flow and resulted in underestimated pollutant concentration in the model.

Blocken et al. (2008) investigated the reliability of steady-state RANS turbulence models for pollutant dispersion simulation through three case studies by comparing with the models with experimental results. They discovered that the turbulent Schmidt number $Sc_t$ has an important influence on pollutant dispersion simulation. The steady-state RANS models underestimated lateral dispersion. Transient simulation was required to achieve more accurate results. Furthermore, unintended, streamwise, high-gradient turbulent kinetic energy was a source of error for the numerical simulation. In most commercial numerical software, the turbulent Schmidt number $Sc_t$ is given as 0.7. Tominaga and Stathopoulos (2007) tested the effect of the Schmidt number $Sc_t$ on flow field simulation since no agreed-upon Schmidt number was used in the simulation. They found out that Schmidt number has significant effect on prediction results of simulation. However, there were still no principles for chosen the Schmidt number $Sc_t$, since it mainly depended on the dominant flow structure.
Tominaga and Stathopoulos (2008, 2011) conducted numerical simulations to uncover discrepancies between the LES model and RANS models. The RANS models underestimated turbulence diffusion near to building surfaces, yet they were very important for predicting pollutant concentration. Li et al. (2008) carried out a LES simulation for street canyon and showed that simulated results were in agreement with experimental data. They concluded that LES is more accurately predicted turbulence intensity than did the RANS model.

2.7 Experiences of real urban case studies

The Project of COST 732 was carried out to assure the quality of micro-models to predict flow and pollutant dispersion in urban and industrial areas (COST 732 model evaluation case studies: Approach and results, COST 732, 2010). Validation data were gained through wind-tunnel model and full-field measurements. CFD simulations were then carried out for comparison. The standard $k−\varepsilon$ turbulence model is mainly for assessment. Several conclusions were reached.

- The measured turbulent kinetic energy ($k$) has certain discrepancies for deviated equilibrium values (Richards and Hoxey, 1993) when it comes to sparsely mounted building elements. It will result in errors for predicting pollutant concentration inside street canyons. Therefore, measured data for vertical wind profile was suggested to be defined at inlet of computational domain in industrial applications.

- RANS turbulence models predicted higher maximum pollutant concentrations close to the source. This was partly due to turbulent kinetic energy error from equilibrium conditions and numerical models, such as the linear diffusion model.

- Sensitivity analysis was also investigated. It mainly focused on the influence of parking areas, source models, building geometries and positions. The results revealed that the source model has no large influence on pollutant distribution around the source. However, parking areas and upwind building geometry have a strong influence on predicting pollutant concentration at the ground level for street canyons.

Systematic analysis was done for the exercise. However, there was still no agreement on preferred turbulence model. Modeling differences were not clearly demonstrated in the evaluated works. Grid analysis for the domain was also not investigated.
Chapter 3 Rough-wall boundary layer flow

3.1 Classification of rough-wall boundary layer flow

The flow retarded by mounted buildings roughness in the bottom of atmospheric boundary layer is defined as roughness surface layer. Since surface layer is the central theme of this work, qualitative description of surface layer and quantitative representation of flow parameters will be briefly given. This layer is close to ground and accounts for approximately 10% of the total atmospheric boundary layer (Roth, 2000). The velocity in the layer is characterized by a high velocity gradient. It is also accompanied by high turbulence levels. Typically, the surface layer is classified into three regions: Inertial, rough-wall and canopy, as shown as Figure 3.1 (Poggi et al., 2004; Fisher et al., 2005).

![Figure 3.1 Three length scales within surface sublayers: Inertial region, rough wall region, and canopy region](Poggi et al., 2004, edit)

**Canopy region**

Perry et al. (1969) proposed a concept called ‘$k$’ and ‘$d$’ type rough-wall flow. When buildings are densely located, the flow below displacement height $d$ will no longer be disturbed by external flow. This is called ‘$d$’ type rough-wall flow. Poggi et al. (2004) also showed that in the canopy region, flow is dominated by energetic motions that are controlled by length scales that reflect the local canopy geometry, such as building height $H$. This occurs regardless of building density and relative height. Normally, the aerodynamic parameter, or zero-displacement height $d$, is introduced to represent the canopy height. However, if buildings are sparsely placed on the ground, there will be no canopy flow, and $d$ will approach zero.

**Rough-wall region**

Above the displacement height, flow is strongly influenced by surrounding buildings and roof elements at building height level, which is termed ‘$k$’ type rough-wall flow (Perry et al., 1969). The layer is overlap of canopy and inertial regions. Poggi et al. (2004) noted that density of surrounding roughness elements is an appropriate scaling parameter to describe the shift between rough-wall and canopy flows in the atmospheric boundary layer. However, the
explicit effect of surrounding buildings on flow is still a challenge in urban areas (Kastner-Klein and Rotach, 2004).

Raupach et al. (1991) proposed the concept of a height of roughness sub-layer $z^*$ in the range of $[2z_H, 5z_H]$ in a real urban region. In this concept, $z_H$ denotes average height of roughness elements (buildings). The exact height of the roughness sub-layer in urban areas is still very controversial (Roth, 2000). In the building-roof layer level to $z^*$, flow is continuously adjusting to ever changing surface roughness, and will never reach to equilibrium. Turbulence analysis demonstrated that the rough-wall region has high horizontal variability and therefore becomes fully three-dimensional (Rotach, 1993; Fisher et al., 2005).

**Inertial region**

The inertial region is above rough-wall region. This inertial region is also known as the *logarithm layer* or *constant turbulence flux layer*. In this region, surface inhomogeneity is negligible. Length scales can be reflected by Monin-Obukhov similarity, where length scales of the vortex are proportional to $z - d$. The height of the inertial region ranges from layer $z^*$ to $0.1\delta$, where $\delta$ is the height of the atmospheric boundary layer (Feddersen, 2005). Turbulence flux in the region is constant in the vertical direction. Aerodynamic parameter, or roughness length ($z_0$), is introduced to describe the velocity and turbulence profiles in the vertical direction. The convenience of introducing roughness length ($z_0$) lies in negligible, detailed, ground roughness elements. However, this method is unreliable to describe sparsely placed buildings.

### 3.2 Mean flow structure

As stated above, flow structure inside street canyons and in the vicinity of building height level is dominated by the length scale of buildings and density of surrounding buildings. Even Perry et al. (1969) classified flow regimes into ‘$k$’ and ‘$d$’ types. Only ‘$d$’ type flow has precise analytic solutions from the ground to top of inertial region. This is known as *logarithm law*. Similar to sand-grain roughness height ($k_s$), two aerodynamic parameters, called *roughness length* ($z_0$) and *zero-displacement* ($d$), are introduced to present main velocity flow in the inertial region. In the rough-wall region, velocity in the vicinity of building roof level significantly interferes with buildings, therefore revealing highly horizontal variability. The mean velocity profile $U(z)$ given by logarithm law displays different extent deviations from the real profile. This is particularly so in the vicinity of building roof height, depending on building density.

In the inertial wall region turbulence flux maintains constant. Velocity scaling called *friction velocity* $u_*$ is

$$u_* = \sqrt{-u'u'}$$

(3.1)

Based on Prandtl’s mixing-length model (Bredberg, 2001), oncoming flow close to the ground level in the inertial region is more accurately ruled by logarithm law. This is also based on field-test and wind-tunnel experiments (Richards and Hoxey, 1993).

$$U(z) = \frac{u_*}{z_0} \ln \left( \frac{z-d}{z_0} \right)$$

(3.2)
In rural or suburban areas, the urban canopy layer disappears and zero displacement height is close to zero. In rural areas, if the height of ABL (δ) and corresponding velocity \( U(\delta) \), or any reference height and velocity, such as meteorology stationary measurement point at 10m height are known, the friction velocity is

\[
 u_* = U(\delta) \times \frac{\kappa}{\ln \left( \frac{\delta - d}{z_0} \right)} \tag{3.3}
\]

Table 3.1 shows typical roughness heights for different region, as well as turbulence characteristics (ASCE Manuals and reports on Engineering Practice No.67, 1999). From rural areas to urban cities, as roughness length increases, turbulence intensity in vertical height increased, as well as the height of atmospheric boundary layer.

<table>
<thead>
<tr>
<th>Class</th>
<th>Terrain Description</th>
<th>( z_0 (\text{m}) )</th>
<th>( \alpha )</th>
<th>( I_u (%) )</th>
<th>( \delta (\text{m}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Open sea, fetch at least 5km</td>
<td>0.0002</td>
<td>0.1</td>
<td>9.2</td>
<td>215</td>
</tr>
<tr>
<td>2</td>
<td>Mud flats, snow; no vegetation, no obstacles</td>
<td>0.05</td>
<td>0.13</td>
<td>13.2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Open flat terrain; grass, few isolated obstacles</td>
<td>0.03</td>
<td>0.15</td>
<td>17.2</td>
<td>275</td>
</tr>
<tr>
<td>4</td>
<td>Low crops; occasional large obstacles,</td>
<td>0.10</td>
<td>0.18</td>
<td>21.7</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>High crops; scattered obstacles, residential suburbs</td>
<td>0.25</td>
<td>0.22</td>
<td>27.1</td>
<td>370</td>
</tr>
<tr>
<td>6</td>
<td>Parkland, bushes; numerous obstacles</td>
<td>0.5</td>
<td>0.29</td>
<td>33.4</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>Regular large obstacle cover-age (dense spacing of low building, forest)</td>
<td>1.0–2.0</td>
<td>0.33</td>
<td>43.4</td>
<td>460</td>
</tr>
<tr>
<td>8</td>
<td>City center with high-and low-rise buildings</td>
<td>( \geq 2.0 )</td>
<td>0.40–0.67</td>
<td>-</td>
<td></td>
</tr>
</tbody>
</table>

As shown in Table 3.1, wind profile and corresponding turbulence intensity is relatively precise in rural or less roughness regions, and can be directly used for wind-induced engineering applications. However in extremely rough urban center more precise methodology is need to estimate zero-displacement and roughness length. Corresponding turbulence characteristics of vertical wind profile must also be provided for simulation.

In urban areas, several proposed empirical equations of aerodynamic parameters are derived from analysis of surface forms. Generally, two methods are used to derive vertical wind-velocity profile: Morphometric and micrometeorological methods (Grimmond and Oke, 1999). The later approach requires tall towers and instrumentation that is expensive and almost impossible for the field site. The morphometric approach is based on wind-tunnel experimental results for simple arrays of roughness elements. Although this method has disadvantages without complex test conditions, it is still reasonably validated by experiment tests and has great values for engineering applications.

Sensitivity analysis was also investigated for three different morphometric approaches (Grimmond and Oke, 1999): height-based (Garratt, 1992; Raupach et al., 1991), height and plan areal fraction (Rotach, 1994; Bottema, 1995b), and height, plan areal density and frontal area density (Raupach, 1994; Macdonald et al., 1998; Bottema, 1995c, 1997). The maximum value of roughness length is close to 0.1\( \bar{H} \) when frontal areal density is in the range of 0.2 to
0.4. Increasing frontal-area density roughness length will decrease to zero. An intermediate building density has the highest roughness length. However, zero-displacement height continues to increase with increasing frontal-area density. From the viewpoint of simplicity and reliability, approaches proposed by Macdonald et al. (1998) and Raupach (1994) are recommended. Kastner-Klein and Rotach (2004) carried out a wind-tunnel experiment on a scaled urban landscape that is focused on inhomogeneous situations. An empirical morphometric formula to derive zero-displacement height and roughness length for an urban city was proposed. Following are details of the morphometric model in this study.

For low-density areas, the following simplified empirical equation was proposed for a height-based approach since a peak showed when plan areal density is in the range of 0.2 to 0.4 (Grimmond and Oke, 1999):

\[
\text{Zero-displacement } d = f_d \bar{H} \\
\text{Roughness length } z_0 = f_0 \bar{H}
\]

(3.4) (3.5)

where \( \bar{H} \) represents average building height, and empirical coefficients \( f_d \) and \( f_0 \) mostly solve as 0.7 and 0.1. If plan-areal density is out of range, the roughness length will be overestimated.

Macdonald et al. (1998) proposed another morphometric method to estimate zero-displacement \( (d) \) for a mean wind profile based on average building height \( \bar{H} \) and plan areal fraction \( \lambda_p = \frac{A_{plan}}{A_{total}} \), valid for all ranges of local plan area density.

\[
d = 1.0 + A^{-\lambda_p}(\lambda_p - 1.0)
\]

(3.6)

Empirical coefficient \( A \) is 4.43 for staggered arrays and 3.59 for square arrays, based on wind-tunnel testing. In real urban cities, value 4.43 is mostly used. \( A_p \) is average plan area of roughness elements and \( A_T \) is total surface area. Building geometry is defined in Figure 3.2.

Accordingly, an empirical model to predict roughness length is expressed as:

\[
\frac{z_0}{\bar{H}} = (1 - \frac{d}{\bar{H}}) \exp\left( -0.5B \frac{C_D}{\kappa} (1 - \frac{d}{\bar{H}}) \lambda_F \right)^{-0.5}
\]

(3.7)
where $B$ is an empirical coefficient (1.0), $C_D$ is a drag coefficient (1.2), $\lambda_F$ is frontal areal density defined as $\lambda_F = \frac{A_F}{A_T}$. These empirical models are more suitable for more homogenous surface situations.

The following empirical morphometric formula was proposed based on experimental data to derive zero-displacement height and roughness length for real urban landscape (Kastner-Klein and Rotach, 2004)

$$\frac{a}{H} = 0.4\lambda_p \exp\{-2.2(\lambda_p - 1.0)\} + 0.6\lambda_p$$

$$\frac{z_0}{H} = 0.072\lambda_p \left[\exp\{-2.2(\lambda_p - 1.0)\} - 1.0\right]$$

Figure 3.3 shows the difference of roughness length and zero-displacement height prediction, respectively based on the GO model (Grimmond and Oke, 1999) and KR model (Kastner-Klein and Rotach, 2004). It confirmed aforementioned statements and also showed that different morphometric methods mainly have significant differences at low frontal areal density, or for areas of medium building density. The maximum values of roughness length predicted by two different methods also varied by approximately 100% which is the major drawback of the morphometric method. In the current study, the morphometric model was used to describe the oncoming wind profile, as physical buildings have identical heights that can be perceived as homogeneous surface.

The weakness of the empirical model is that it is mostly based on flat and homogeneous surfaces in high-density built areas. Turbulence properties are measured in the constant flux layer. The more close to the roof level of urban areas, the greater the uncertainty of the proposed empirical model, which will become largely attributed to the three-dimensional effect at building roof level. As a result the success of these methods highly depends on how well UBL turbulence can be described by empirical relationships derived over flat and homogeneous surfaces (Roth, 2000).
3.3 Turbulence structure

At neutral stratified conditions, a logarithm law wind profile can be transformed as:

\[ I_u = \sqrt{\frac{u'^2}{U(z)}} = \frac{k}{\ln\left(\frac{z}{z_0}\right)} \cdot \sqrt{\frac{u'^2}{u_*}} \quad (3.10) \]

Turbulence intensity \( I_u = \sqrt{\frac{u'^2}{U(z)}} \) is an important parameter to represent turbulence level at different vertical heights. Furthermore, turbulent kinetic energy \( k = \frac{1}{2} (u'^2 + v'^2 + w'^2) \) can then be derived as:

\[ k = \frac{1}{2} (u'^2 + v'^2 + w'^2) = \frac{u_*}{2} \sum (\sigma_i/u_*)^2 \quad (3.11) \]

According to the above equations, the turbulence parameters, friction velocity \( u_* \), and standard deviation of velocity \( \sigma_i/u_* \) present turbulence profile of oncoming wind.

**Standard deviation of velocity \( \sigma_i/u_* \)**

Wind-tunnel experiments discovered that the value of the normalized standard deviation of vertical velocity \( \sigma_i/u_* \) maintains constant in the inertial region for neutral atmospheric boundary flow (Cermak and Cochran, 1994; Roth, 2000). Different field data or experimental data are shown in Table 3.2.

<table>
<thead>
<tr>
<th>Studies</th>
<th>Location</th>
<th>( \sqrt{u'^2}/u_* )</th>
<th>( \sqrt{v'^2}/u_* )</th>
<th>( \sqrt{w'^2}/u_* )</th>
<th>( k/u_*^2 ) (average data)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Roth (2000)</td>
<td>Urban</td>
<td>2.4</td>
<td>1.91</td>
<td>1.27</td>
<td>5.51</td>
</tr>
<tr>
<td>Rotach (1995)</td>
<td>Urban (close to roof)</td>
<td>1.69</td>
<td>1.53</td>
<td>0.92</td>
<td>3.09</td>
</tr>
<tr>
<td>ESDU (1985)</td>
<td>Rural</td>
<td>2.75</td>
<td>2.14</td>
<td>1.51</td>
<td>7.26</td>
</tr>
<tr>
<td>Panofsky and Dutton (1984)</td>
<td>References</td>
<td>2.39±0.03</td>
<td>1.92±0.05</td>
<td>1.25±0.03</td>
<td>5.48</td>
</tr>
<tr>
<td>Counihan (1975)</td>
<td>Rural</td>
<td>2.5</td>
<td>1.9</td>
<td>1.25</td>
<td>5.71</td>
</tr>
</tbody>
</table>

Counihan (1975) reviewed normalized standard deviation of velocity profiles mainly in rural or less rough areas in near-neutral conditions from field or experimental test. Roughness length \( z_0 \) ranged from 0.0001 to 5.0m. However, most collected roughness length is below 0.1. That means the proposed empirical equations are more valuable for suburban or rural areas. Counihan (1975) concluded that constant values of \( \sigma_i/u_* \) are only valid for the constant shear stress layer. Standard deviations of velocity variance \( \sigma_i/u_* \) in different coordinates are respectively given as 2.5, 1.875 and 1.25. ESDU (1985) and Panofsky and Dutton (1984) researched the same conclusions. Roth (2000) reviewed over 50 field measurements for
normalized standard deviation of velocity in urban areas. In the constant shear street layer in urban areas, average values of $\sigma_i/\bar{u}_*$ were respectively approximately 2.4, 1.91 and 1.27.

When the flow continually decreased to the vicinity of building height, where wind flow is highly influenced by ground buildings, Rotach (1995) showed that wind speed and turbulence profile highly depend on atmospheric stability conditions. At near-neutral runs, longitudinal velocity variance $\sigma_u/\bar{u}_*$ is nearly close to a constant value of about 1.75 for the entire vertical height. For lateral velocity variance ($\sigma_v/\bar{u}_*$), the numerical value is slightly reduced to approximately 1.55. Even in near-neutral conditions, it tends to decrease with height. Vertical velocity variance ($\sigma_w/\bar{u}_*$) is about 0.9 in the vicinity of the building roof level.

**Friction velocity ($u_*$)**

Cermak and Cochran (1994) discovered that in rural areas, friction velocity $u_*$ in the vertical direction maintains constant to $0.1\delta$, where $\delta$ represents the height of atmospheric boundary layer. Therefore, relative Reynolds stress plotted in Figure 3.4 will be restricted to a height lower than $0.1\delta$. Counihan (1975) collected meteorological statistical data between Reynolds stress and roughness length from field tests or wind-tunnel as illustrated in Figure 3.4. Reynolds stress increased as roughness length in rural areas increased. The following empirical model was proposed to present the relationship between friction velocity and roughness length based on Equation (3.12).

$$\frac{u_*}{\bar{u}_0} = 2.75 \times 10^{-3} + 6 \times 10^{-4} \log z_0$$ (3.12)

![Figure 3.4 Relationship between Reynolds stress and roughness length in rural area](image)

Unlike rural or less rough areas, turbulence structure in urban surface layers will significantly affected by inhomogeneous nature of underlying surfaces and high roughness lengths. Due to variation of wind direction across cities, roughness lengths will also have large differences. As a result, vertical wind profiles and turbulence structures will not only be influenced by high roughness length, but also by directional variation, especially in the vicinity of building roof level.
As stated above, friction velocity can be estimated by logarithm law with conditions of given reference heights and reference velocities. At building level, maximum friction velocity $u_{\text{max}}$ is used to estimate local friction velocity $u_{l}$. Based on experiments (Rotach, 2001), $z^*$ equals 1.5~2$z_H$ fitting the experimental results best in urban environment. According to Fisher et al., 2005, the following equation can be used to predict local friction velocity in urban rough wall regions:

$$\left(\frac{u_{l}}{u_{\text{max}}^2}\right)^b = \sin\left(\frac{\pi}{2}Z\right)^a \quad \text{when} \quad Z \leq 1$$

(3.13)

where $u_{l}$ is local scaling velocity, $u_{l}^2 = -u'w(z)'$, and $Z = \frac{z - d}{z^*} - d$ is a non-dimensional height. The constant $a$ and $b$ are respectively set as 1.28 and 3.0. The problem of with the method is that it is difficult to estimate maximum friction velocity $u_{\text{max}}$ in urban environments. It is closely relevant to local roughness circumstances. As a result, correctly estimating turbulence properties inside the rough-wall region is still a challenge. This is another factor that will result in uncertainty of flow and pollutant dispersion inside roughness surface layer for industrial applications.
Chapter 4 Literature review on Wind Flow and Pollutant Dispersion inside Street Canyons

4.1 Definition of idealized street canyons

Street canyons define streets with buildings in parallel (Nicholson, 1975). An isolated street canyon can be defined by two parameters: Aspect ratio ($AR$) $H/W$ and length-to-width ratio $L/H$. Dimensions of $H$, $W$ and $L$ respectively represent flat roof height in vertical direction, width of street, and length of upwind buildings in streamwise and spanwise directions. Depending on the ($H/W$), street canyons can be shallow ($H/W < 0.5$), uniform ($H/W = 1.0$) or deep ($H/W > 2.0$). According to ($L/H$), street canyon can be short ($L/H < 3.0$), medium or long ($L/H > 7.0$). In terms of relative height between neighboring buildings, street canyons can be classified into three configurations: step-down, step-up, and even (Figure 4.1). The ratio of relative height $H_1/H_2$, is used to describe these configurations. Street canyons without identical heights are asymmetric. Those street canyons with identical heights are symmetric (Ahmad et al., 2005).

![Figure 4.1 Definition of street canyon](image)

4.2 Flow characteristics inside idealized street canyons

4.2.1 Perpendicular wind direction

When ambient wind direction is perpendicular to uniform street canyons, wind flow regimes inside street canyons are classified in terms of $AR$ ($H/W$): isolated roughness flow (IRF), wake interference flow (WIF) and skimming flow (SF). These are shown in Figure 4.2. The critical values for the flow regime transition in two dimensions are shown as Table 4.1. Oke’s (1988) simulation results only considered wind effect without temperature. However, Sini et al. (1996) and Baik and Kim (1999) added solar radiation and surface temperature effects.
Figure 4.2 Perpendicular flow regimes inside street canyons with different H/W aspect ratios (Ahamed et al., 2005). (a) Isolated roughness flow; (b) Wake interference flow; (c) Skimming flow

Table 4.1 Critical street canyon aspect ratio (H/W) for flow regime transition (Li, et al., 2006)

<table>
<thead>
<tr>
<th>Transition</th>
<th>IRF – WIF</th>
<th>WIF-SF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Oke (1988)</td>
<td>0.3</td>
<td>0.7</td>
</tr>
<tr>
<td>Sini et al. (1996)</td>
<td>0.11-0.125</td>
<td>0.67</td>
</tr>
<tr>
<td>Baik and Kim (1999)</td>
<td>-</td>
<td>0.5-1</td>
</tr>
</tbody>
</table>

Since corner recirculation flow and cross-canyon flow co-exist around a single building in a real environment, Hunter et al. (1992) investigated flow transition metrics impacted by upstream building geometry. As street canyon ratio (L/H) increases from 1 to 7, corner recirculation will accordingly rise from 0.8H to 1.5H. Therefore, the AR of H/W for flow transition from SF to WIF will alter from 1.25 to 0.67. The length of cross-canyon flow will also increase from 1.8H to 4.6H. Therefore, the metric ratio of H/W for flow transition from WIF to IRF will alter from 0.55 to 0.22.

Much research has focused on SF regimes impacted by wind and temperature effects. Uehara and Murakami (2000) used a stratified wind tunnel to investigate the effect of atmospheric stability (mainly thermal stratification) on flow in and above street canyons. The bulk Richardson number ($R_b$) defines atmospheric stability in street canyons as:

$$R_b = \frac{gH(T_H - T_0)}{((T + 273)(U_H)^2)}$$

(4.1)

The $R_b$ will be greater only if wind flow velocity at building height is small or the temperature gradient inside the street canyon is large, which means more stability. The experimental results are illustrated as Figure 4.3. Flow recirculation inside street canyons will
become weak with increasing atmospheric stability or $R_b$. Stable atmospheric has negative effect for the downward flow entered into street canyon. Therefore, there are less external flows mixed in the street canyon and carry pollutant out of street canyon. When $R_b$ exceeds 0.4–0.8, flow velocity inside street canyons nearly approaches to zero.

Figure 4.3 Contours of vector flow inside street canyons: (Left) stable case $R_b=0.79$; (Left) Neutral case $R_b=0$; (Left) unstable case $R_b=-0.21$; (Uehara and Murakami, 2000)

Sini et al. (1996) undertook numerical research accounting for temperature and wind effect. The researcher took street $AR$ into account for different heating positions in street canyons to investigate flow patterns. Kim and Baik (1999) also undertook systematic research on heating-position effect on flow patterns inside street canyons using ARs ranging from 0.5 to 3.5. This is depicted in Table 4.2. A heating source located in an upwind building surface of street canyons was more beneficial for enhancing momentum exchange between outside mean flow and flow inside street canyons. Furthermore, Kim and Baik (2001) investigated temperature gradients between ground and inside air effect on flow patterns. The temperature gradients can reinforce flow recirculation inside street canyon. The temperature gradient postponed flow recirculation transition in some extent, for constant $AR$. Furthermore, different circulations formed, depending on $AR$ in the skimming flow regime. This is shown in Table 4.3.

Table 4.2 Flow patterns in street canyons with ARs under different heating conditions (Kim and Baik, 1999)

<table>
<thead>
<tr>
<th>AR</th>
<th>0.5</th>
<th>1</th>
<th>1.5</th>
<th>2</th>
<th>2.5</th>
<th>3</th>
<th>3.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>No</td>
<td>One circulation</td>
<td>Two counter-rotating recirculations</td>
<td>Three vertically aligned, counter-rotating recirculations</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Upwind</td>
<td>One circulation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bottom</td>
<td>Two circulation</td>
<td>Two counter-rotating recirculations in lower layer and one in upper layer</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Downwind</td>
<td>One circulation</td>
<td>Two counter-rotating recirculations</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 4.3 Different flow patterns in skimming flow regime depending on street-canyon AR

<table>
<thead>
<tr>
<th>Flow patterns</th>
<th>One circulation</th>
<th>Two circulations</th>
<th>Three circulations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sini et al. (1996)</td>
<td>0.67 – 1.67</td>
<td>&gt;1.67</td>
<td></td>
</tr>
<tr>
<td>Baik and Kim (1999)</td>
<td>1</td>
<td>1.5-3</td>
<td>3.5</td>
</tr>
<tr>
<td>Huang et al. (2000)</td>
<td>0.25-1</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Kim and Baik (2001)</td>
<td>0.6-1.2</td>
<td>1.4-3.2</td>
<td>3.4-3.6</td>
</tr>
<tr>
<td>Jeong and Andrews (2002)</td>
<td>0.67 – 1.6</td>
<td>1.6-2.67</td>
<td>&gt;2.67</td>
</tr>
</tbody>
</table>

Generally the flow inside street canyons is driven by mean shear layer at roof level. There is a coupling connection between outside flow at roof level and inside street canyons. However when oncoming wind flow velocity is below a threshold, which ranges from 1.5m/s to 1.7m/s for symmetrical street canyons, the connection will disappear (Schatzmann and Leitl, 2011). In this case, temperature will have an important effect upon flow patterns in street canyons. In comparing wind- and temperature-induced flow fields inside street canyons it was found that wind-induced flow recirculation is the main factor, while temperature-induced flow recirculation is secondary. Temperature-induced effect will dominate the flow field inside street canyons only if wind-induced effect is negligible.

4.2.2 Oblique and parallel wind direction

Stathopoulos et al. (1986) investigated wind speed and turbulence intensity influenced by wind direction, height of building, and passage width. Turbulence intensity inside the passage increases to the maximum value when wind direction is perpendicular to the passage. Furthermore, a wider passage has greater turbulence intensity. Stathopoulos (1992) also used the concept of influential scale to examine pedestrian wind conditions influenced by building height, and parallel and sequential buildings. The largest amplification factor for wind speed at pedestrian height occurs at \(y/S = 0.4\).

Blocken et al. (2007) systematically investigated the effect of passage width on flow regimes for parallel passages. This was based on Stathopoulos’ (1992) research. Typical speed conditions around parallel buildings are depicted in Figure 4.4. Three flow regimes are demonstrated, according to metric \(W/S\). Wilson (1979) defined influential length scale \(S\) as:

\[
S = B_s^{0.67} B_L^{0.33}
\]

where \(S\) represents the influential length scale. \(B_s\) is the smaller building dimension, perpendicular to wind direction (m) and \(B_L\) is the larger building dimension, perpendicular to wind direction. If \(B_L\) is larger than eight times \(B_s\), \(B_L\) is replaced with \(8B_s\). When \(W/S\) is smaller than 0.125, a resistance flow regime is initiated. In this situation, wind speed amplification factor inside the center line of parallel passages is not more than the flow amplification factor at the corners of parallel buildings. When \(W/S\) is between 0.125 and 1.25, this is called interaction flow. The flow separations at a single stream corner will interact and merge together into a single jet flow. As a result, the speed amplification factor at center line will be larger than a single corner stream. If \(W/S\) increases to greater than 1.25, there is no
corner stream interaction at parallel passages. The flow around parallel buildings will act as an isolated building.

Few studied have examined wind flow development inside street canyons for oblique wind direction. Ahmad et al. (2003) showed that a helical flow pattern is generated inside street canyons with an oblique wind direction.

4.3 Characteristics of pollutant dispersion inside street canyons

Much focus has been given to the pollution dispersion in street canyons, as concentrations of canyon flow directly affects urban air quality. Many empirical models have been developed to predict pollution distribution in street canyons. Two methods have been investigated to evaluate concentration distribution in street canyons: uniform concentration (which perceives street canyon as boxes) and nonuniform concentration. If the canyon volume is an uniform source field, the variation of concentration in canyon volume related to time is expressed as (Lee and Park, 1994):

$$V \frac{dC}{dt} = -qC$$

(4.3)

where \(V\) is canyon volume, \(C\) is canyon flow concentration, and \(q\) is influx strength into canyon. At steady state, the equation solution is:

$$C(t) = C_0 e^{-t/\tau}$$

(4.4)

Where \(\tau = \frac{V}{q}\) represents constant. At time \(t = \tau\), concentration \(C\) of canyon flow will be decreased by \(e^{-1}\). Hoydysh and Dabberdt (1988) studied the impact of wind direction and canyon configuration on flow characteristics and pollutant dispersion. An empirical exponential function was proposed to describe concentration distribution along vertical height:

$$K = a e^{b \frac{z}{H}}$$

(4.5)

where \(a\) and \(b\) are empirical coefficients, and \(H\) and \(K\) are building height and non-dimensional concentration. Since vehicle-emitted pollution can be perceived as line pollutant source, Rafailidis and Schatzmann (1997) used the non-dimensional form

$$K = \frac{CUH}{Qe}$$

(4.6)
to represent pollutant distribution in street canyons. Here, $C$ represents the measured vol./vol.
ethane trace concentration (in the range of $[0,1]$); $U$ represents reference velocity at reference
eight; $L$ is the length of the line source. $Q_e$ is the trace-gas pollutant strength.

Studies have explored the mechanisms of pollutant transportation in street canyons. Concentrations of canyon volumes are mainly affected by three factors (Wedding et al., 1977; Hoydysh and Dabberdt, 1988; Lee and Park, 1994; Meroney et al., 1997; Kastner-Klein et al., 2000): canyon $AR$ of height to width $AR = H/W$, Reynolds number at roof level $Re_r = \frac{Hu}{v}$, and Peclet number defined by $Pe_r = Re_rPr_r$, where $Pr_r$ represents Prantle number at roof level. Roof-level Reynolds number $Re_r$ is usually related to local turbulence intensity, and $Pe_r$ is related to local thermal effect. Three transfer mechanisms were proposed to characterize wind-induced pollutant transfer (Liu and Barch, 2001; Salizzoni et al., 2009): turbulence diffusion within the street canyon ($\bar{u}_d$), advection due to mean recirculation flow ($\bar{u}_c$), and turbulence dispersion across shear layer at roof level ($\bar{u}_t$). These are shown in Figure 4.5.

![Figure 4.5 Simplified illustration of vehicle-emitted pollutant transfer mechanisms within the street canyons](image)

**Effect of geometry of street canyons with flat roof shape and identical height**

As mentioned previously, street canyon geometry is represented by aspect ratio ($AR$). Flow characteristics and regimes inside street canyons also directly depend on $AR$ and the corresponding number of circulations, as shown in Table 4.1-4.3 in this chapter. These flow regimes, and circulations inside street canyons, have a great effect on pollutant dispersion. The pollutant removal process is a combination of advective transport and turbulence diffusion. As street canyon width increases (decreasing $AR$), the pollutant dispersion depends more on advective transport (Lee and Park, 1994; Sini and Anquetin, 1995). Results also showed with SF, multi-vortex pollutant dispersion will be dominated by turbulence diffusion, while convective effects are negligible. In such cases, vehicle-emitted pollution at canyon bottom is mainly dominated by diffusion from bottom vortex to upper vortex, and pollution is difficult to transfer out of the street canyon. For one-vortex SF, WIF, and IRF wind flows, pollutant dispersion has a weak connection with turbulence diffusion (Ellen et al., 2005) and will have low pollutant concentration inside street canyons. Pollutant concentrations on windward side of street canyons are normally lower than on leeward side.

**Effect of relative height of upstream to downstream buildings ($H_2/H_1$) and roof shape**

Hoydysh and Dabberdt (1988) carried out a field test to study the effect of asymmetric street canyons with flat roofs on kinetic and dispersion characteristics. Wind-tunnel
experiments (Rafailidis and Schatzmann, 1995; Kastner-Klein and Plate, 1999) were also conducted to investigate effect of roof shapes on pollutant distribution inside street canyons. Results showed that concentrations in the step-up notch are lower than for the other two notches, since high mean circulation velocity is generated inside street canyons. Pollutant concentration at the windward surface is higher than at the leeward surface for even and step-up notches. For step-down notches, the concentration distribution reversed. Kastner-Klein et al. (2004) explained the increased pollution level for pitch roofs at upwind buildings. The increase was attributed to a recirculation zone formed at the upwind building ridge and spanning across the downwind building. At the recirculation zone, low wind velocity hampered formation of street-canyon vortices. As a consequence, flow became almost stagnant and pollution was trapped.

Chan et al. (2001) investigated effects of relative height, from 0.5 to 2.0, on pollutant concentration of isolated roughness flow for a two-dimensional street canyon. There is maximum value of pollutant concentration when relative height increases from 0.5 to 2.0. As the blockage effect from the leeward building side increases, the pollutant concentration will tend to increase with relative height. From the viewpoint of efficient dispersion, the street canyons should not be uniform. Variations in roof height provide better ventilation.

Xie et al. (2005) studied roof shapes and building geometry effects on pollutant dispersion inside street canyons. Huang (2007) studied the impact of wedge-shaped roofs on the pollution dispersion, including 17 case studies. Yassin (2011) studied pollution dispersion by comparing different roof shape and heights.

Results showed that, the even and step-up configurations generally have the same pollutant concentration in street canyons, and that concentration on the leeward is greater than on the windward side. The main reason for the variation is attributed to circulation wind speed and number of vortices generated within the canyon. For step-down configuration, ability of pollutant dispersion is smaller than with the even and step-up. Double-eddy or so called counter-vortex appeared in the street canyon. Pollutant concentration on the leeward side was smaller than on the windward side. Vortices at canyon bottoms bring the pollutants from the leeward to the windward side. Furthermore, shear zone at roof level also changes, depending on roof shape.

**Effect of thermal or temperature**

Sini and Anquetin (1995) investigated wall-temperature influence on pollutant dispersion in street canyons. They concluded that wall heating for the leeward wall and ground enhanced flow recirculation, which improved pollutant dispersion. However, solar wall heating for the windward wall had less effect on flow recirculation and pollutant dispersion. Kim and Baik (2001) investigated the influence of $H/W$ on pollutant dispersion by thermal effect. Only 10 degrees of temperature difference has a visible effect on the pollutant dispersion in street canyons for windward-wall heating. Xie et al. (2005) also investigated thermal effect on pollutant dispersion in street canyons and reached similar findings.

**Effect of neutral-external turbulence flow**
Different studies (Lee and Park, 1994; Salizzoni et al., 2009) demonstrated that external flow has nonlinear interaction with locally generated vortices inside street canyons and directly influences turbulence intensity when $AR=1$. The study also showed that, for SF regime, pollutant transportation out of street canyons is largely constrained by dispersion across the shear zone at roof level. However, in real environments, pollutant dispersion demonstrated different characteristics between suburban and urban areas. Recirculation eddies only intermittently formed in suburban street canyons. Pollution was then transferred out through intermittent air. Permanent eddies formed inside urban street canyons due to shear zone at roof level which trapped pollutants (Meroney et al., 1997).

The upstream turbulence flow divides into two parts: turbulent flux and mean flow. Numerical studies (Bai and Kim, 2002; Li et al., 2009) revealed that pollution dispersion out of the canyon is mainly forced by the turbulence flux, while mean flow forces pollutants back into canyons. Numerical sensitivity research investigated the effect of turbulence intensity on pollutant dispersion in street canyon. Increasing turbulence intensity at roof level enhanced vortices inside street canyons, which improved pollutant transportation (Baik and Kim, 2003).

**Effect of small-scale surface roughness elements**

In urban boundary layer, roughness elements are demonstrated as various length scales, which can be divided into two groups: Large-scale elements, such as buildings and small-scale elements, such as façades, and roofs. A wind-tunnel study investigated the effect of these length-scale elements on flow field and pollutant dispersion for two-dimensional street canyons (Salizzoni et al., 2009). Results showed that small-scale roughness enforced turbulence intensity and momentum transfer when the street canyon $AR$ was greater than 1, and has relatively little effect for street canyons with small $AR$s. That is due to pollutant dispersion at high $AR$s being driven by turbulence transport at roof level. Length scale elements below $H−d$ are not taken into consideration in physical models.

**Effect of trees and vehicle motion**

Trees are often planted along urban streets. The effect of trees on nature ventilation and pollutant dispersion in street canyons was investigated through wind-tunnel experiments (Mochida et al., 2006; Gromke et al., 2007a, 2007b, 2009). Figure 4.6 shows concentration distribution on the leeward and windward sides of street canyons when wind flow is perpendicular to street canyon with $AR$ at 0.5. (Available at: [www.ifh.uni-karlsruhe.de/science/aerodyn/CODASC.htm](http://www.ifh.uni-karlsruhe.de/science/aerodyn/CODASC.htm)).

Gromke et al. noted that there is a critical value of tree porosity. Below a certain threshold, there is no change in pollutant concentration inside street canyons by the presence of trees. When wind direction is changing, there is a greater tree-induced pollutant trapping effect inside street canyon with high a $AR$. Therefore, dominant wind direction should be considered when planning tree planting. These findings are also confirmed by a study based on experimental data and a real urban case (Buccolieri et al., 2011).
Ahmad et al. (2005) reviewed studies of the effect of vehicle motion on pollutant dispersion inside street canyons. Main contributions for the study came from Kastner-Klein et al. (1998, 1999, 2000). Three different traffic arrangements were studied. Pollutant concentrations inside street canyons are showed in Figure 4.7. In a one-way traffic arrangement, vehicle motion markedly changed turbulence transportation along street canyon axes. Therefore, pollutant concentration inside street canyon was also altered along street canyon axes. In a two-way traffic arrangement, concentration distribution on both leeward and windward surfaces is slightly changed, compared to the reference case. Vehicle-induced turbulence counteracted in opposite directions. Results also revealed that concentration inside street canyon decreased with increased ratio of vehicle- to wind- speed and increased vehicle density.
Chapter 5 Summary of research and discussion

As mentioned in Chapter 4, pollutant dispersion inside street canyons is driven by three mechanisms: turbulence diffusion, advection due to flow recirculation, turbulence diffusion across the shear layer at roof level. Turbulent diffusion research has been widely investigated. Advection due to flow recirculation also investigated, such as vehicle motion induced lateral flow and pollutant dispersion, are also demonstrated in the literature.

The work mainly focuses on flow and pollutant concentration inside street canyons that is induced by turbulence diffusion across shear layer at roof level. The contribution of the work includes two aspects. First, a novel numerical model is established to investigate flow and pollutant concentration inside street canyons (Scenario study A). Factors influencing the reliability of results are systematically investigated. The model can directly use for practical applications. In the Scenario study B, the impact of roof shapes on flow and pollutant concentration inside street canyons are studied. The mechanism of roof shape impact on pollutant dispersion is explicitly demonstrated. The results are valuable for building shape optimization in urban building design.

5.1 Scenario study A

Three issues will be discussed to numerically investigate air flow characteristics and pollutant concentrations inside street canyons. First, relatively reliable inlet wind profiles, including wind speed and turbulence properties, are used as input for CFD model. Second, it is necessary to choose a reliable turbulence model, a relevant numerical method, and grid generation. Third, model and research limitations will also be illustrated in this section. Based on validated numerical models, the impact of surrounding buildings and street canyon geometry on flow and pollutant dispersion are investigated. Conclusions based on simulation results are also discussed.

5.1.1 Methodology

As described as Chapter 3, wind environment in suburban or rural areas is quite simple and can be expressed as a relatively reliable wind profile with input for flow and pollutant dispersion simulation. This paper reviewed data by Counihan (1975) and ASCE for wind profiles in suburban or rural areas. These data can be directly used, since flow is stable in neutral conditions and there is no significant interference with surface roughness elements. The morphometric method is used to generate inlet profiles to describe oncoming wind profiles in urban case studies. However, as stated by Grimmond and Oke (1999), the morphometric method is unable to perform flow inside rough-wall regions, especially if the ground surface is highly inhomogeneity. Therefore, therefore, two-dimensional street canyons with flat roofs are investigated in this research.

As illustrated in Chapter 2, numerical methodology is used for flow and pollutant dispersion. It is already widely used in wind engineering application. However, constant values, such as $C_\mu$ and $S_{ct}$ of turbulence models, are waiting for further development. The LES model is an appropriate model to predict wind profiles and pollutant concentration. Nevertheless, time-variable inputs for wind profile are still challenging. In addition, wind profiles are strongly sensitive to variation in wind direction. Therefore, RANS models are still
chosen as tools for simulating flow and pollutant dispersion. Relevant wall function problems are also discussed in this thesis. Proper turbulence models, and relevant boundary conditions, and numerical method are chosen through research.

Pollutant source is perceived as a line source, due to vehicle motion inside street canyons. However vehicle moving induced turbulence is not taking into account in this paper. Normally, vehicle pollutants comprise a range of particle sizes and chemical reaction. In the current study, the size of pollutant source only is limited to a range where the source can be considered a gas, and there is no chemical reaction. Therefore, it is a passive source. Whole modeling framework is shown in Figure 5.1.

5.1.2 Physical model and oncoming wind profile analysis

In order to quantitatively illustrate the uncertainty of the morphometric method, aerodynamic parameters, including roughness length ($z_0$) and zero-displacement height ($d$) for ideal two-dimensional street canyons are predicted. Ideal experimental street geometry consists of two-dimensional square bars (0.06m×0.06m) that are parallel at both sides. The spacing between the bars could be varied in order to represent upstream building density. An investigation was performed for objected street canyon with four ARs (0.2, 0.5, 1 and 2), which corresponded to three flow patterns: IRF, WIF, and SF. In order to study the impact of
oncoming plan-area density on pollutant dispersion in street canyons, four building types were chosen, as depicted in Figure 5.2. Spacing between bars for upstream fetch was also designated as 0.2, 0.5, 1 and 2. Pollutant source was released at the middle ground levels of street canyon.

![Figure 5.2 Physical model of scenario study](image)

Table 5.1 Estimation of $z_0$, d and $u_*/U_{ref}$ for empirical models and experimental data

<table>
<thead>
<tr>
<th>Models</th>
<th>Parameters</th>
<th>Unit</th>
<th>Fetch configuration ($H/W$)</th>
<th>0.2</th>
<th>0.5</th>
<th>1</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Empirical model (Kastner-Klein and Rotach, 2004)</td>
<td>$z_0$</td>
<td>mm</td>
<td></td>
<td>3.79</td>
<td>4.8</td>
<td>4.33</td>
<td>3.11</td>
</tr>
<tr>
<td></td>
<td>d</td>
<td>mm</td>
<td></td>
<td>31.1</td>
<td>46.5</td>
<td>54</td>
<td>57.3</td>
</tr>
<tr>
<td></td>
<td>$u_*/U_{ref}$</td>
<td>m/s</td>
<td></td>
<td>0.08</td>
<td>0.086</td>
<td>0.088</td>
<td>0.082</td>
</tr>
<tr>
<td>Empirical model (Grimmond and Oke, 1999)</td>
<td>$z_0$</td>
<td>mm</td>
<td></td>
<td>7.53</td>
<td>5.83</td>
<td>3.07</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>d</td>
<td>mm</td>
<td></td>
<td>16.4</td>
<td>35.4</td>
<td>45.7</td>
<td>57.3</td>
</tr>
<tr>
<td></td>
<td>$u_*/U_{ref}$</td>
<td>m/s</td>
<td></td>
<td>0.097</td>
<td>0.091</td>
<td>0.08</td>
<td>0.067</td>
</tr>
<tr>
<td>Experimental data (Salizzoni, 2008)</td>
<td>$z_0$</td>
<td>mm</td>
<td></td>
<td>-</td>
<td>1.70</td>
<td>0.31</td>
<td>0.13</td>
</tr>
<tr>
<td></td>
<td>d</td>
<td>mm</td>
<td></td>
<td>-</td>
<td>52</td>
<td>57</td>
<td>59</td>
</tr>
<tr>
<td></td>
<td>$u_*/U_{ref}$</td>
<td>m/s</td>
<td></td>
<td>-</td>
<td>0.061</td>
<td>0.049</td>
<td>0.045</td>
</tr>
<tr>
<td>Experimental data (Rafailidis, 1997)</td>
<td>$z_0$</td>
<td>mm</td>
<td></td>
<td>-</td>
<td>-</td>
<td>0.4</td>
<td>0.6</td>
</tr>
<tr>
<td></td>
<td>d</td>
<td>mm</td>
<td></td>
<td>-</td>
<td>-</td>
<td>55</td>
<td>60</td>
</tr>
<tr>
<td></td>
<td>$u_*/U_{ref}$</td>
<td>m/s</td>
<td></td>
<td>-</td>
<td>-</td>
<td>0.06</td>
<td>0.055</td>
</tr>
<tr>
<td>Experimental data (Meroney, 1996)</td>
<td>$z_0$</td>
<td>mm</td>
<td></td>
<td>3.00</td>
<td>1.00</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>d</td>
<td>mm</td>
<td></td>
<td>45</td>
<td>50</td>
<td>57</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>$u_*/U_{ref}$</td>
<td>m/s</td>
<td></td>
<td>0.100</td>
<td>0.075</td>
<td>0.050</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 5.1 shows calculated values of roughness length ($z_0$), zero-displacement height ($d$) and non-dimensional friction velocity ($u_*/U_{ref}$) with empirical morphometric models, as well as comparison data of wind-tunnel experiments measured by other researchers. As stated above, peak values of roughness length appeared when the ratio of height to width was in the range of 0.2 to 0.4 in the scenario study. As the ratio of height to width ($H/W$) continues to increase, roughness length $z_0$ will decrease. Experimental and empirical data showed the same trend.

However, empirical models predict a higher value of roughness length than do experimental data, while zero-displacement heights are highly in agreement. As non-dimensional friction velocity was more sensitive with roughness length prediction than with zero-displacement height, the results of non-dimensional friction velocity also revealed higher values compared to experimental data.

### 5.1.3 Numerical model validation
In order to validate pollutant distribution in street canyons, Rafailidis and Schatzmann (1997) carried out wind tunnel experiments in the atmospheric boundary layer wind tunnel of the Meteorological Institute of Hamburg University. A scale of 1:500 was used. The pollutant concentrations were represented into non-dimensional form as \( K = \frac{C U_{\text{free}} H L}{Q_e} \). \( C \) represents the measured vol./vol. ethane-trace concentration (in the range of \([0,1]\)). \( U_{\text{free}} \) represents reference velocity at a height of 0.5 above the tunnel floor, at 5m/s. \( L \) is the length of line source 0.9. \( Q_e \) is the trace gas strength. The tracer line source gas with mixture of ethane and air was emitted from source point with a strength of \( Q_e = 4l/h, Q_a = 100l/h \).

In the numerical scenario study, the computational domain is defined as \( L \times Z = (4B + 5H + W) \times 6H \). In the horizontal direction, the approaching fetch and outflow fetch extend respectively five and nine times street width. In the vertical direction, the block height is five times street height above the ground. In the simulation, the source boundary condition set the value of velocity inlet as 0.0025m/s to ensure little interruption between pollutant source and main street flow (Meroney, 1996). The scalar boundary conditions in other situations are set with zero diffusive flux conditions.

The ambient wind direction is assumed to be perpendicular to the isolated street canyon. A passive pollutant point source was assumed to be released at center ground-level of the two-dimensional street canyon. The spacing between the bars could be varied to represent upstream building density. Combining morphometric, source and turbulence models, and this model is then validated with wind-tunnel data. Grid, turbulence models are also tested before application simulation. The effect of street canyon geometry, wind speed and upcoming wind profiles on flow fields and pollutant concentrations inside street canyon are then investigated.

**Grid-independent analysis**

To ensure results are grid-independent and investigate proper grid meshing, the current research carried out grid analysis with same boundary conditions and turbulence model. Seven grid meshes are shown in Figure 5.2. During simulation, the roughness of the building surface takes into account. Roughness height of building surface was set as 0.0006m, which is 1/1000 of building height \( H \). The default standard \( k – \varepsilon \) turbulence model with standard wall function in Fluent 14.0 is used in the study. The expansion ratio for all meshing is set at 1.1. Details about the meshing are shown in Table 5.2.

<table>
<thead>
<tr>
<th>Meshing</th>
<th>First grid height</th>
<th>Node</th>
<th>Total cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top-wall</td>
<td>Leeward</td>
<td>Windward</td>
<td>Ground</td>
</tr>
<tr>
<td>Mesh1</td>
<td>0.006</td>
<td>0.006</td>
<td>0.006</td>
</tr>
<tr>
<td>Mesh2</td>
<td>0.006</td>
<td>0.006</td>
<td>0.006</td>
</tr>
<tr>
<td>Mesh3</td>
<td>0.004</td>
<td>0.006</td>
<td>0.006</td>
</tr>
<tr>
<td>Mesh4</td>
<td>0.006</td>
<td>0.002</td>
<td>0.006</td>
</tr>
<tr>
<td>Mesh5</td>
<td>0.006</td>
<td>0.002</td>
<td>0.002</td>
</tr>
<tr>
<td>Mesh6</td>
<td>0.006</td>
<td>0.002</td>
<td>0.006</td>
</tr>
</tbody>
</table>
The testing procedure was conducted as followings: First, 20 nodes were vertically distributed in street canyons. The standard $k - \varepsilon$ turbulence model and wall function were used. First grid height of all wall surfaces for reference mesh sets to 0.006 in order to put $y^+$ of first grid height in logarithm region. Then maintaining first grid height for all wall surfaces, double refining reference mesh achieves to Mesh1. Mesh1, Mesh2, Mesh4 and Mesh5 were achieved by changing the first grid height at wall surface and ground surface to ensure $y^+$ located approximately 1.

Simulation results for all cases are shown in Figure 5.4. In comparison to experimental results (Rafailidis and Schatzmann, 1997), the reference mesh underestimated pollutant dispersion at the leeward surface. Mesh1 showed no significant improvement reproducing pollutant distribution inside street canyons, even with double refinement. High grid density at the center of street canyons was not a key parameter to affect pollutant distribution simulation. Mesh4 was closest to the experimental data. This means high grid density at ground level is a key for reliable prediction of pollutant distribution inside street canyons. However increasing grid density at windward surface has negative effect for the pollutant distribution prediction by comparing mesh4 and mesh5.

It is widely known that constant value $C_\mu$ at the near wall region for standard $k - \varepsilon$ turbulence model is unreliable. Flow will not be satisfactorily simulated inside street canyons. However, pollutant concentrations for both windward and leeward surfaces are significantly consistent with experimental data. Gorle et al. (2010) also noted this conflict. They pointed out that satisfactory prediction of the velocity field is not a sufficient condition for correctly predicting the concentration field. Future challenge is how to simulate flow and pollutant concentration fields at the same time.

Mesh3 tests the importance of grid density at the leeward side. In comparing Mesh3 and Mesh4, it was found when grid density at the leeward surface increases, pollutant concentration is closer to experimental data. Finally, Mesh6 is designed to ensure that simulation results are grid-independent.

Based on above analysis, the following conclusions can be made. A high gradient of pollutant concentration is generated at ground level and leeward region. The roughness of wall surface is significantly reduced by using scaling. It is easy to set the first grid $y^+$ at approximately 1. Therefore, a high grid density is needed to capture the gradient in order to precisely reproduce the concentration and velocity fields. As seen in Figure 5.5, pollutant concentrations at both leeward and windward surfaces are consistence between simulation results with standard $k - \varepsilon$ turbulence model and experiment. Therefore, Mesh4 is the optimized meshing for further investigation. However, it is controversial whether standard $k - \varepsilon$ turbulence model is the proper model to simulate flow and pollutant dispersion inside street canyons. Following section demonstrates results of pollutant distribution simulated by various turbulence models.
Figure 5.3 Graphs of seven meshing
Mesh4 is used to test effect of RANS turbulence models on pollutant distribution inside street canyons. Three turbulence models, standard $k-\varepsilon$, RNG $k-\varepsilon$ and realizable $k-\varepsilon$ turbulence model, were investigated with the same boundary conditions. As seen in Figure 5.6, RNG $k-\varepsilon$ and realizable $k-\varepsilon$ turbulence models highly predicted pollutant concentration at both leeward and windward surfaces of street canyons, compared to standard $k-\varepsilon$ turbulence model. The conclusions agreed with other research results (Hanna et al., 2004; Milliez and Carissimo, 2007). Results simulated by RNG $k-\varepsilon$ turbulence model agreed with experimental data near the bottom of windward surface. However, standard $k-\varepsilon$ turbulence
model was still selected in this paper since it was more consistent with experimental data on the whole.

Figure 5.7 shows turbulent kinetic energy at inlet and outlet, simulated by three different turbulence models. At a height greater than twice the building height, results for turbulent kinetic energy were significantly consistent with the inlet profile. However, results were scattered at roof level. This may have attributed for pollutant distribution differences for the three models.

Franke (2006) suggested that standard $k - \varepsilon$ turbulence models should not be used for wind engineering problems. The RSM turbulence model could be interesting model to use in the future. Currently, however, very few turbulence models have been widely validated (Tominaga et al., 2008).
Comparison was made between $k - \omega$, SST $k - \omega$, and standard $k - \varepsilon$ turbulence models. The reason for this model comparison is that it is easier to set the first grid $y^+$ close to 1 for wind-tunnel scale model. Under such circumstances, the near-wall region can be directly solved by a low-Reynolds turbulence model without wall function. For the inlet profile, the specific dissipation rate is defined as $\omega = k/\beta^*\varepsilon$. Relevant data can be directly derived from $k - \varepsilon$ model. Other boundary conditions and source pollutant are similarly defined with $k - \varepsilon$ models. Figure 5.8 shows results of pollutant concentrations at both windward and leeward surfaces for different models.

Results revealed that pollutant concentration predicted by standard $k - \omega$ and SST $k - \omega$ models are higher than results simulated by standard $k - \varepsilon$ model. The deviation between measurement data and results predicted by $k - \omega$ model was over more than 50%. Figure 5.9 shows vertical turbulent kinetic energy profile at inlet and outlet predicted by three different models. Standard $k - \omega$ simulated severe deviation between inlet and outlet. Standard $k - \omega$ failed to capture flow structure at roof level since there was high gradient shear stress. SST $k - \omega$ revealed better results than standard $k - \omega$ near roof level. For the distant building roof domain, turbulent kinetic energy agreed with reference inlet profile since transformed $k - \varepsilon$ was used.

Unfortunately flow field data inside street canyons was not available, and there was no flow field comparison between different turbulence models. Results simulated by standard $k - \varepsilon$ showed better agreement with measurement data than $k - \omega$ models. However we cannot say if standard $k - \varepsilon$ turbulence model is better than $k - \omega$ models for simulation of atmospheric boundary flow. Conclusions can only be reached if a systematic comparison between flow data and dispersion data is done. In the real urban environment, there is a high gradient of turbulence kinetic energy in the vicinity of roof. $k - \omega$ turbulence models have potential to capture the turbulence gradient at roof level. And SST $k - \omega$ will have better potential for modeling flow and pollutant dispersion than standard $k - \omega$.

![Figure 5.8 Non-dimensional pollutant concentration $K$ for three RANS turbulence models](image)
In real environments, length scale is greater than wind-tunnel scale. Therefore full-scale investigation of pollutant distribution is essential. Since a 1:500 scale model was used for validation, the full scale enlarges by 500. Wall roughness height becomes 0.03 compared to wind-tunnel scale. Oncoming wind speed is still set to 5m/s at height of 250m, which is consistence with wind-tunnel settings. The first grid at wall surface is 0.035 to fulfill the numerical model requirements that the first grid height needs to exceed the physical roughness height. A 40×40 grid is used for all street canyons. A 80×400 grid is set for vertical and horizontal direction above canyon. Reference length $L_{ref}$ is set at 2H to estimate Reynolds numbers for two models. Reference wind velocity $U_{ref}$ at specific heights for full and wind-tunnel models respectively are 2m/s and 3.7m/s (Values from simulated results). The Reynolds numbers for the two models are:

$$Re_{full} = 3.8 \times 10^6 \quad \text{and} \quad Re_{wind-tunnel} = 28,280$$

Figure 5.10 shows pollutant distributions at windward and leeward surfaces of street canyon for wind-tunnel and experimental model data. The full-scale model predicted high pollutant concentrations than did the wind-tunnel model. As seen in Figure 5.11, the first grid $y^+$ at ground surface is close to 1 for wind-tunnel model, the wall function was not involved in simulation. In the full-scale model, $y^+$ at ground surface is in the logarithm law region. The turbulence model use wall function for simulation. The uncertainty of the wall function is attributed to errors between small-scale and full-scale models. This is also reflected by difference results between Mesh and Mesh4 since $y^+$ of the first wall grid at ground surface is also in the logarithm region.

The situation in real urban environments is worse since high roughness elements are existed in the canopy layer. The physical models need to simplify for practical application. These simplify is reflected by high surface roughness height of buildings. The first grid $y^+$ value is no longer in logarithm region (Hargreaves and Wright, 2007). Wall function may
need further development to reliably predict flow and pollutant transportation in the roughness surface layer. Small roughness elements for full-scale models are ignored in small-scale models. This shows that wind-tunnel validation data needs to carefully be distinguished when reproducing pollutant dispersion in real environments. It also brings into question how detailed physical roughness needs to be modeled to ensure reliable simulation results in real urban environments. In order to take into account the effect of small-scale obstacles, a canopy model needed to develop in future work. Extra terms are needed to add to the basic $k - \varepsilon$ turbulence model to decrease wind velocity but increase turbulence. If a low Reynolds number turbulence model is used, such a canopy model should also be developed.

However, even Reynolds number for the full-scale model is a factor approximately 135 times greater than the small-scale model. Results of pollutant concentration at both leeward and windward surfaces still reasonably agree with each other. This is due to viscous effect in the RANS models, which is strongly suppressed by the factor $1/Re$. At sufficiently high Reynolds numbers, the introduced Reynolds effect for flow and scalar transportation could be negligible. In this circumstance, flow and scalar transportation inside street canyons becomes Reynolds-number-independent. However, if velocity falls below certain threshold, the Reynolds effect could dominate flow patterns and scalar transportation inside street canyons.

Figure 5.10 Non-dimensional pollutant concentration $K$ for two length scales

(Rafailidis and Schatzmann, 1997)
Analysis of Schmidt number $S_{ct}$

Schmidt number $S_{ct}$ is an important factor that affects pollution diffusion inside street canyons. However, there are no consent principles to select Schmidt number $S_{ct}$ for specific flow conditions. We investigated the impact of Schmidt number $S_{ct}$ on pollutant distribution inside street canyons in the current model. The default value was 0.7 in Fluent, with 0.4 and 1.0 as simulated and comparison results as shown in Figure 5.12. Grid Mesh4 with wind-tunnel profile (Rafailidis and Schatzmann, 1997) was tested in the simulation, based on grid analysis in the above section. As Schmidt number increased from 0.4 to 1.0, non-dimensional pollutant concentration at both leeward and windward surfaces of street canyons also increased. This is consistent with Blocken et al. (2008). For windward surface, results agree more with measurement data when $S_{ct}$ is in the range of 0.7 to 1.0. However, for leeward surface, $S_{ct}$ setting to approximately 0.7 is preferable. For this investigation, setting $S_{ct}$ to 0.7 is an acceptable option.
Based on the above analysis, standard $k - \varepsilon$ turbulence model with the default Schmidt number was used.

**Comparison between simulation and experimental data**

One of the aims was to investigate the impact of surrounding buildings density on pollutant distribution inside street canyons. Due to lack of measurement data, the morphometric method was applied to predict the inlet vertical profile, which was used for numerical simulation. Two situations were predicted for $H/W = 1$ and $H/W = 2$. Results (Figure 5.13) showed that, for street canyon $H/W = 1$, only one vortex was generated. Main recirculation flow carried the pollutants to the leeward surface of the canyon, then shifted out of the canyon. Therefore, there was higher pollutant concentration on the leeward than on the windward surface. However, two vertical counter-vortices were generated in the street canyon with $H/W = 2$. One was located at the ground level and another at upper level. Pollutants were trapped at bottom of street canyon and were only transported by turbulence diffusion. Therefore, higher pollutant concentration was seen at both surfaces of the street canyon. On the other hand, for a street canyon with $H/W = 1$, numerical simulation underestimated pollutant concentration at leeward and windward surfaces. This is attributed to overestimation of roughness length for the oncoming wind profile (Table 5.1), thereby overestimating turbulent kinetic energy. Nevertheless, pollutant distribution at both surfaces agrees well with measurement data.

![Figure 5.13 Non-dimensional pollutant concentration comparison with simulation and experiment at windward and leeward surfaces of street canyons for $H/W=1$ (left) and $H/W=2$ (right)](image)

**5.1.4 Numerical simulation results and discussions**

**Flow characteristics in street canyons**

The wind flows are depicted in Figure 5.14 for four different ratios of $W/H$. The ratios were set at 2, 1, 0.5, and 0.2. Four upwind fetches representing four vertical wind velocity and turbulent profiles were also used to investigate the impact of upcoming wind profile on flow patterns in street canyons. Simulation results showed that flow patterns were independent of upcoming wind profiles, by just relying on street-canyon geometry. Three typical wind flow patterns were shown: SF, WIF, and IRF (Oke, 1988). For $W/H = 2$, two vertical counter-vortices were seen (Figure 5.14a). For $W/H = 1$, one main vortex was generated, accompanied by two small vortices in the bottom corners of the street (Figure 5.14b). As
\( W/H \) decreased, flow characteristics shifted from one main vortex (Figure 5.14c) to two horizontal counter-vortices (Figure 5.14d).

![Flow patterns](image)

Figure 5.14 Flow patterns for street canyon values of \( W/H \): (a) 2, (b) 1, (c) 0.5, and (d) 0.2

**Wind speed effect**

Hoydysh et al. (1974) suggested that Reynolds number \( (Re = \frac{U_{ref}H}{v}) \), based on free stream velocity \( (U_{ref}) \) and building height \( (H) \), should exceed 12,000 to ensure that the flow pattern is independent of wall viscous effects (Meroney et al., 1996). In the current study, the Reynolds number was approximately 20,000. The impact of free stream wind speeds on pollutant distribution was investigated for two cases: \( W/H = 1 \) and \( W/H = 2 \). Upcoming wind velocities, 3m/s, 5m/s and 7m/s, were chosen. Simulation results of non-dimensional pollutant distribution at leeward and windward surfaces are shown in Figure 5.15. As shown in results for both \( W/H = 1 \) and \( W/H = 2 \), as free stream velocity \( (U_{ref}) \) increased, non-dimensional pollutant concentrations at leeward and windward surfaces decreased. High free stream wind velocity was therefore better for transporting pollutant source out of street canyons. In case of stronger wind speeds, the shear flow region extended deeper into the street canyon and there was enhanced pollutant dispersion. As mentioned before, flow patterns inside street canyons are independent of upcoming wind profiles. Pollutant concentrations at both surfaces only varied quantitatively.
Building-plan-area density and canyon-geometry effect

In order to quantitatively investigate the impact of surrounding buildings and street-canyon geometry on pollutant concentrations, for four upwind fetches, plan-area density ($\lambda_f$) varied 0.67, 0.5, 0.33, and 0.167. Results are shown in Figure 5.16. Each subfigure includes four different target street canyons corresponding to H/W of 2, 1, 0.5, and 0.2.

In Figure 5.16a, non-dimensional pollutant concentration at leeward and windward surfaces decreased with decreasing value $H/W$ for upwind fetch. Street-canyon geometry determined the flow patterns, which determined pollutant transportation out of street canyons. With multi-vortices, pollutant dispersion was mainly attributed to turbulence diffusion, which will suppress pollutant trapped in the canyon. This was the reason for significantly high pollutant concentrations for street canyons when $H/W = 2$. As $H/W$ decreased, pollutant dispersion became dominated by advection, which was more efficient at transporting pollutant (Lee and Park, 1994). For $H/W$ at 0.2, two horizontal, counter-rotating vortices were generated, pollutants were then carried to leeward surface and transported directly out of the street canyon. While at the windward surface, pollutants was isolated by two counter-rotating vortices, which accounted the low pollutant distribution. Need to point out that position of pollutant source in such a situation will significantly change distribution of pollutant inside street canyons (Meroney, 1996). Figure 5.16b-d shows that the same mechanism occurred for all upwind fetch.

Comparisons of simulation results for four upwind fetches are shown in Figure 5.16. Two important conclusions can be reached. First, for one vortex generated in the street canyons with $H/W$ equal to 1, 0.5, and 0.2, the average non-pollutant concentration on the leeward surface street canyon decreased from 120 to 75. Windward surface decreased from 25 to 15, which is a decline of almost 50%. As stated before, coarse surrounding buildings with high, upcoming turbulent intensity benefited for pollutant dispersion. However, for $H/W$ at 2, pollutant concentration near the bottom of street canyons has no obvious influence from surrounding buildings. The collection between external and internal flow is fully lost. This means pollutant dispersion is dominated by local geometry of the street canyon itself. Meanwhile, improved surrounding buildings planning can improve pollutant transportation.
Figure 5.16 Non-dimensional pollutant concentrations at windward and leeward surfaces of street canyons for different upwind plan area densities: (a) $\lambda_f = 0.67$, (b) $\lambda_f = 0.5$, (c) $\lambda_f = 0.33$, and (d) $\lambda_f = 0.167$

### 5.1.5 Conclusions and discussion

The paper established a numerical model, combined with a morphometric method and computational fluid dynamics (CFD) model to predict flow characteristics and vehicle pollutant distribution in street canyons. Analysis was done to test the effect of grid, Reynolds number, and turbulence models on pollutant concentrations. Potential improvements of simulation results introduced by numerical model are composed from the following aspects:

- Through sensitivity analysis of grid density, we discovered that height distribution of 30 to 50 grid points for the reference building height fulfills simulation requirements. This study chose 40 grid points for the reference building height. A high grid density needs at bottom ground surface to capture pollutant concentration gradients. In real urban environment, because of high roughness elements, grid meshing still needs to be investigated to ensure that results are grid-independent.

- $k - \varepsilon$ turbulence models have a well-known disadvantage. The models are unreliable for capturing near-wall flow. It is also incapable of highly flow separation and rotation situations. However, pollutant concentrations inside street canyons at leeward and windward surfaces were consistent with measurement data in the current study. It proved satisfactory prediction of velocity field is not a sufficient condition for correctly predicting
concentration field. Simultaneously achieving reliable flow field and pollutant concentration inside street canyons by advanced turbulence models still needs further investigation.

- At small-scale sizes, low Reynolds number models, such as series of $k - \omega$ turbulence models, can be directly used for the simulation model without a wall function. This is potentially better for flow field simulation. The series of $k - \omega$ is also capable of capturing shear stress layer in the vicinity of building roof level. In the circumstance, SST $k - \omega$ showed greater ability than standard $k - \omega$ for simulating flow and pollutant dispersion. However, further research is needed as to what extent full-size model must be scaled to maintain Reynolds number independence. Furthermore, advanced turbulence model, such as canyon models also need to consider the effect of small-scale obstacles.

- Oncoming wind profiles for practical application are still the main challenge for the simulation of flow and pollutant dispersion in urban environments. Morphometric methods introduce numerous errors into simulation results. One possible solution is to enlarge simulation domain sacrificed by simulation time and resources.

The validated model is then used to investigate influence of wind speed and plan building areal density and street-canyon geometry on flow pattern and pollutant distribution. Simulation results showed that flow patterns is mainly depended on street-canyon geometry. Pollutant dispersion is closely related to flow characteristics. If building construction is dense, pollutant sources will be trapped inside the canopy level and result in high pollutant concentrations. Furthermore, proper plan areal density of building can improve pollutant dispersion. Therefore, a reasonable design of upwind building density will benefit pollutant dispersion in the main roads of urban cities. It is also noted that higher oncoming wind speed is beneficial for pollutant source transported out of street canyon.

This study was limited by only looking at homogenous buildings. However, turbulence structures in real urban surface layers will significantly be affected by inhomogeneous underlying surfaces and high roughness lengths. Due to variation of wind direction, roughness lengths will also show large differences. As a result, vertical wind profiles and turbulence structures also be influenced by wind directional, especially near building roof levels. The Weakness of empirical models is that, as with morphometric methods, they are mostly based on flat and homogeneous surfaces in high-density areas. Turbulence properties are measured in the constant flux layer. The closer to roof level in urban areas, the more uncertainty the empirical model becomes. This is largely attributed to the three-dimensional effect. As a result, the success of these methods greatly depends on how well UBL turbulence can be described by empirical relationships derived over flat and homogeneous surfaces (Roth, 2000).

Additionally, flow patterns and pollutant dispersion in urban cities will be significantly influenced by atmospheric stability induced from solar radiation. Since established models are only applicable in near-neutral conditions, further developments should consider atmospheric stability in the mathematical model. Wind-tunnel validation for complex, three-dimensional flow patterns, pollutant dispersion, and GIS-integrated methodologies for urban city prediction should also become further research topics.

5.2 Scenario study B
We demonstrated that main challenge for wind and pollutant dispersion research is still how to illustrate wind profiles close to building roof levels. In that layer, flow becomes three-dimensional, complex, and subject to strong variation from building shapes. Therefore, this section presents a quantitative study of the impact of roof shapes on flow characteristics and relevant pollutant concentrations inside street canyon.

A two-dimensional street canyon with different roof shapes and configurations in rural area is ideal to investigate the effect of roof shapes on flow field and pollutant dispersion. Therefore, it can greatly ignore interference from upcoming wind profiles. The ambient wind direction was assumed to be perpendicular to the isolated street canyon (AR=1). A passive pollutant source was assumed to be released at center ground level of the street canyon. Four roof shapes (flat, left-wedge, slanted, and right-wedge) and three typical building configurations (even, step-down and step-up) were considered. The simulation results were initially validated with wind-tunnel experimental data. Thereafter, the validated model was used to simulate other physical cases.

5.2.1 Methodology
5.2.1.1 Street canyon configuration

Figure 5.17 shows two-dimensional street canyon configurations, which consists of three types of sequential parallel streets: even, step-down and step-up. In order to simulate constant vehicle exhaust emissions, line pollutant source S was located at center ground level the street canyon. For each configuration, four categories of roof shape (flat, left-wedge, slanted, and right-wedge) for upwind buildings were simulated. All roofs had the same height, while the peak places were in different positions. All tested streets were cubic blocks with same geometry. The canyon had the width B and height H, with AR $\frac{B}{H} = 1$. The ratio of roof height to building height $H_r / H$ was 0.33. The wind direction was perpendicular to the upstream buildings with velocity $U_{free} = 5m/s$. 

Figure 5.17 Street canyon configurations: (a) even; (b) step-down; (c) step-up; (d) canyon geometry
5.2.1.2 Grid, boundary conditions and numerical method

Figure 5.18 Grid and computational domain

All the boundary model values were directly taken from Meroney (1996) to validate the computational model. The experiment was implemented in the atmospheric boundary layer wind tunnel of the Meteorological Institute at Hamburg University. The scale was 1:500 for an open-country situation. Blocks replicating surface roughness were placed on the floor in a regular array in a staggered pattern. Building blocks with flat roofs represented street canyons. The square-cross section has a height to width of $H \times B = 0.06m \times 0.06m$. The tracer line source gas (mixture of ethane and air) was emitted from source point with a strength of $Q_e = 4l/h$, $Q_a = 100l/h$. A free-stream wind approach velocity at a height of 0.65 above ground flow was defined as 5m/s.

Shown in Figure 5.18, the computational domain size was $L \times H = 1.02m \times 0.42m$ for the 12 cases. The approaching domain extends five times the building height and nine times the building height for outflow fetch in the horizontal direction. 40 points were assigned for reference building height $H$. The block height was set to seven times the building height above ground level in the vertical direction. The first grid height was approximately 0.002 for wind-tunnel scale, and the domain included 60,000 cells. Boundary conditions were set as follows: the horizontal inlet velocity profile was set as power law with power index 0.143. Accordingly, the roughness length ($z_0$) of upcoming ground was defined as 0.0015m. Using Equation 2.18, friction velocity $u_*$ for upcoming flow is estimated as 0.34. Turbulent kinetic energy $k$ and dissipation rate of incoming flow in equilibrium conditions followed the equations $k = \frac{u_*^2}{\sqrt{C_\mu}}$ and $\varepsilon = \frac{u_*^3}{\kappa(z+z_0)}$, which were also assumed to be in equilibrium. $u_*$ represented friction velocity, $\kappa$ was Von Karman constant of 0.41; $C_\mu$ was turbulence model constant of 0.09. The vertical velocity of the inlet profile was zero. At ground level, standard wall function was used. Roughness height for approaching ground surface is set as 0.0015 to match inlet profile. Roughness height of building surface is set as 0.00006, which is 1/1000 of the reference building height. At top boundary, a velocity-like inlet boundary profile was given (Blocken et al., 2007). Exit boundary conditions were set with outflow. In the simulation, the source boundary condition set value of velocity-inlet as 0.0025m/s to ensure no interruption with main street flow. Mass ratio of ethane to air was 0.04. Other conditions were defined as:
Table 5.3 Numerical model settings

<table>
<thead>
<tr>
<th>Items</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Software version</td>
<td>Ansys Fluent 14.0</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>RNG $ k - \varepsilon $ model with constants $ C_{1e} = 1.42, C_{2e} = 1.68, C_\mu = 0.0845, \sigma_k = 1.0, \sigma_\varepsilon = 1.3, S_c \epsilon = 0.7 $</td>
</tr>
<tr>
<td>Inlet boundary</td>
<td>$ U(z) = \frac{u}{u^<em>} \left[ \ln \left( \frac{z+z_0}{z_0} \right) \right], k = \frac{u^</em>^2}{\sqrt{\psi}}, \varepsilon = \frac{u^*^3}{\psi(z+z_0)} $</td>
</tr>
<tr>
<td>Outlet boundary</td>
<td>Outflow $ \frac{\partial}{\partial x} (u, v, w, k, \varepsilon) = 0 $</td>
</tr>
<tr>
<td>Upper boundary</td>
<td>Constant values of velocity, turbulent kinetic energy and turbulent dissipation rate extracted from inlet profile</td>
</tr>
<tr>
<td>Ground boundary</td>
<td>Standard wall function</td>
</tr>
<tr>
<td>Wall boundary</td>
<td>Standard wall function</td>
</tr>
<tr>
<td>Discretized method</td>
<td>Finite volume method</td>
</tr>
<tr>
<td>Numerical method</td>
<td>Second-order upwind for all differential equations</td>
</tr>
<tr>
<td>Convergent criteria</td>
<td>$ \left</td>
</tr>
</tbody>
</table>

**Model validation**

According to the experiment results, the source strength had negligible effect on pollutant distribution. A sampling system with 20 measurement points mounted in the street canyon parallel to street axis collected in individual gas samples in inflatable balloons. The pollutant concentrations measured were represented into dimensionless form $ K (K = \frac{CUHL}{Q_e}) $, where $ C $ represented the measured vol./vol. ethane trace concentration (in the range of $ [0,1] $); $ U $ represented reference velocity at a height of 0.065 above tunnel floor. $ L $ was the length of line source 0.9. $ Q_e $ was the trace gas strength.

Figure 5.19 The contours of velocity fields for isolated street canyon(a), concentration and measured non-dimensional concentration $ K $ (b) at isolated street canyon surfaces

Figure 5.19(b) shows the measured and simulated dimensionless pollutant concentrations for reference canyon a1 for leeward surface of the upwind building and
windward surface of the downwind building. The simulated data for the windward side greatly agreed with measured data. Furthermore, as airflow separated at roof level of upstream buildings, the pollutant was transported by airflow to the upstream roof. This completely agreed with the experiment data by Meroney (1996). On both wall sides, the error between simulated and measured data was less than 20%. As seen from the data comparison, the trend of dimensionless pollutant concentration for both leeward and windward sides decreased with height in the vertical direction. The rate of decreasing concentration on the leeward side was much steeper than on the windward side.

5.2.2 Simulation results and discussion

Since pollutant dispersion inside street canyons mainly depends on flow pattern and turbulence intensity at building height level, this section will present simulation results of flow patterns inside and at roof level of the street canyons, turbulence intensity of the canopy at building height, and non-dimensional pollutant concentration distribution on the leeward and windward surfaces of the street canyons.

5.2.2.1 Flow characteristics in street canyons

Figure 5.20 shows the simulated velocity vectors within isolated street canyons with four roof shapes for three configurations. For reference case b1, there was a main clockwise vortex in the street canyon. At the bottom corner of canyon street, there were two weak vortices. At the roof of the upwind building, airflow was separated from the left corner point to the left roof corner of the downwind building. The magnitude of velocity inside the street canyon was less than 0.5. For the even configuration of street canyon (b1, b2, b3, b4), just as with reference case b1, only one main clockwise vortex was generated for different roof shapes of upstream buildings. The results agreed well with Oke (1988). However, there was a high airflow separation at roof height of left-wedge roof shapes of upstream buildings, the vortex center of vortex was in a more upward position than for the other three roof shapes.
For the step-down configuration (a1, a2, a3, and a4), flow separation, reattachment points and vortices generated inside street canyon were unlike each other because of high upstream buildings with different roof shapes. The situations were worse with flat and slanted roof. Significant second vortices formed inside street canyons and another vortex was generated at the upper region. The reattachment point for both cases was closed to the right roof corner of downstream buildings. Street canyons with left and right wedge roof shape only has one main vortex showed formed.

For the step-up configuration of street canyon (c1, c2, c3 and c4), flow regimes is similar to even configuration. However, the centre of vortice were raised up compared to even configuration. A second, weak vortice formed at the corner of ground level.

As a whole, flow patterns in street canyons were mainly affected by building configurations. The impact of roof shape on flow regimes inside street canyon was more significant for the step-down configuration.

5.2.2.2 Turbulence intensity and pollutant concentrations inside street canyon

The horizontal direction is normalized relative height, from leeward surface to windward surface with building height $H$. Figure 5.21a shows the turbulence intensity at normalized height 1.0 with building height $H$. Figure 5.21b shows the non-dimensional pollutant concentration at windward and leeward surfaces inside street canyons. Figure 5.22 and Figure 5.23 have same settings.

Step-down configuration

As seen in Figure 5.21a, simulated results revealed that roof shapes of upstream buildings dramatically influenced turbulence intensity of downward flow inside street canyon for all three configurations. Left-wedge roof generated the strongest turbulence intensity at reference height level, among the four roof shapes for all three configurations. In other three roof shapes, the slanted roof generated the least turbulence intensity. The right-wedge and flat
roofs had comparative number. The relative difference of turbulence intensity among different roof shapes was greater than 50%.

This had significant influence on pollutant distribution inside street canyons. As seen in Figure 5.21b, the street canyon with left-wedge roof has the lowest pollutant concentration at leeward and windward surfaces. This is corresponding to the highest turbulent intensity at building height level. Similarly, street canyon with slanted roof has the highest pollutant concentration due to lowest turbulence intensity at building height level. For flat and slanted roofs, concentrations at leeward surface are greater than windward surface due to double counter-rotating vortices form inside street canyon.

![Figure 5.21](image1.png)

Figure 5.21 Graph of turbulence intensity (a) and pollutant concentration (b) for step-down

Even configuration
As seen in Figure 5.22a, left-wedge roof generates the highest turbulence intensity at reference height level, among the four roof shapes for all three configurations. Follows are slanted and flat roofs. The right-wedge roof generates the lowest turbulence intensity. The relative difference of turbulence intensity among different roof shapes was greater than 40%.

The connection between turbulence intensity and pollutant concentration shows in Figure 5.22b. The street canyon with left-wedge roof has lowest pollutant concentration at windward and lewards surfaces. The street canyon with right-wedge roof has the highest pollutant concentration at windward and leeward surfaces. The distributions are directly corresponded to the turbulence intensity at building height level. The concentration at windward surfaces are higher than leeward surfaces due to single vortex generating in the street canyon for all roof shapes.

![Graph of turbulence intensity and pollutant concentration](image-url)
Step-up configuration

As seen in Figure 5.23a, turbulence intensities at building heigh level for all roofs are similar to even configuration qualitatively. Quantitatively to say, the absolute values of turbulence intensities for step-up are 20% greater than even configuration. The relative difference of turbulence intensity among different roof shapes was greater than 50%.

The street canyon with left-wedge roof has lowest pollutant concentration at windward and leewards surfaces. The street canyon with right-wedge roof has the highest pollutant concentration at windward and leeward surfaces. The concentration at windward surfaces are higher than leeward surfaces due to single vortex generating in the street canyon for all roof shapes.

Figure 5.23 Graph of turbulence intensity (a) and pollutant concentration (b) for step-up
As seen in Figure 5.21a, 5.22a, and 5.23a, pollutant dispersion inside canyon street was strongly impacted by roof shape for three configurations and highly related to flow regimes. Left-wedge roof generated the strongest turbulence intensity at reference height level, among the four roof shapes for all three configurations. The right-wedge roof had the least turbulence intensity at reference height level for even and step-up configurations. The slanted roof generated the least turbulence intensity for the step-down configuration at reference level. Slanted and flat roofs had similar turbulence intensities at reference level for even and step-up configurations.

As seen in Figure 5.21b, 5.22b, and 5.23b, left-wedge roof shapes was more effective than other three roof shapes for all configurations. Slanted and right-wedge roof shapes were less effective for transporting pollutant out of street canyons. For even and step-up configurations, the right-wedge roof shape was less effective than other three roof shapes for pollutant transportation out of street canyon. This was directly attributed to turbulence intensity at roof level.

Additionally, non-dimensional pollutant concentrations inside street canyons are close to each other, for even and step-up configurations with different roof shapes of upstream building. Both configurations showed a greater ability than step-down for the pollutant dispersion. Furthermore, pollutant concentration at the windward surface was greater than at the leeward surface since flow carried pollutants toward the windward side. Interestingly, pollutant concentrations inside street canyons for the step-up configuration were slightly less than even configuration. This was mainly attributed to a blocking effect of downstream buildings forcing oncoming fresh flow re-entrainment into street canyons.

5.2.3 Conclusion and discussions

It concluded that, for even and step-up configurations, there was only one main clockwise vortex generated in street canyons and the non-dimensional pollutant concentration at the windward surface was much smaller than at the leeward surface. With step-down configuration, two counter-clockwise vortices formed inside and upward of street canyons with flat and slanted roof shapes of upstream buildings. Building configurations dominated the flow regime and pollutant transportation inside street canyons. To some extent, even and step-up configuration had an equal ability to remove pollutants out of street canyons.

According to the results, building roof shapes had a significant impact on flow separation at the leading corners of the upstream buildings (especially for step-down configuration). The main effects impacted of roof shapes were turbulence intensity depth and strength of shear layer inside street canyons. Roof shapes slightly changed velocity inside street canyons. Both of these were important for removing pollutant out of street canyons.

The pollutant dispersion inside street canyons can be attributed to two indicators: flow patterns inside street canyons and turbulence intensity at roof level. Left-wedge roof generated the strongest turbulence intensity for all configurations. The results also proved that it had lowest pollutant concentration at windward and leeward surfaces for all configurations. Right-wedge roof should avoid in even and step-up configurations.
Buildings are densely built in urban environments. They are interfered each other. It is hard to separate incoming wind flow and targeted street canyons. Therefore, it is challenge to investigate the impact of roof shapes on pollutant dispersion in urban circumstances. Site-dependent case studies could investigate in further research.
Chapter 6 Conclusions and future work

Conclusions and future work take the form of answers to the research questions. Based on the literature review and current research, we have answered questions proposed in the introductory chapter:

What is the state-of-art of knowledge of flow patterns and pollutant dispersion characteristics inside street canyons?

When wind flow direction is perpendicular to street canyons, flow patterns inside street canyons can be divided into three types: isolated roughness flow (IRF), wake interference flow (WIF), and skimming flow (SF). Flow regimes inside street canyons mainly depend on aspect ratios (ARs) of street canyons. In neutral conditions, isolated roughness flow will exist inside street canyons when AR is below approximately 0.3. Virtually, two horizontal counter-rotating vortices are generated inside street canyons. By increasing AR, the flow regime will transfer from IRF into WIF. If the AR is greater than 0.7, SF regimes be evident. When AR is 1.5, multiple, vertically counter-rotating vortices will form inside street canyons.

Pollutant dispersion has a strong relationship with flow regimes inside street canyons. According to literature review and investigation, pollutant dispersion is dominated by advection due to flow recirculation inside street canyons with IRF and WIF flow regimes. With an SF regime and only one vortex, pollutant dispersion is dominated by turbulence diffusion across the shear layer at the roof layer. Quantitative analysis showed that pollutant concentrations at the leeward surface of street canyons are normally three or four times higher than at the windward surface with identical roof heights. Furthermore, when there are vertical, multi-rotating vortices inside street canyons, pollutant dispersion will be dominated by turbulence diffusion. In this situation, pollutants are trapped deep inside street canyons.

Pollutant concentrations inside street canyons are significantly influenced by vortices, vehicle motion and trees along street canyon axes. Researchers showed that vehicle motion strongly changed pollutant concentrations at both and enhanced transportation of pollution to surrounding buildings due to lateral diffusion vortices. Trees mainly constrained external fresh air flowing into street canyons, thereby reducing pollutant dilution. However, when tree porosity was reduced to a critical threshold, the effect became negligible. Trees also altered lateral flow when wind direction was oblique in a stream-wise direction.

How can location of street canyons and surrounding effects be integrated into mathematical modeling of air quality prediction? Does density of built-up areas influence flow fields and pollutant dispersion inside objected street canyons?

As stated in the previous chapter, flow recirculation inside street canyons is driven by external flow at roof level. External flow is entrained into street canyons, and then diluted with pollution. Through turbulence diffusion and advection, pollutants are then transported out of street canyons. In rural and suburban areas, external flow can be relatively precise, as illustrated by the empirical model measured from the field test. By integrating the empirical model into the numerical model, we can then evaluate surrounding effects on flow fields and pollutant characteristics inside street canyons. As the roughness of surrounding roughness
elements increases, turbulence kinetic energy also increases. It is then benefited for pollutant source transported out of street canyon.

However, external wind flows in urban cities near roof levels are complex and strongly influenced by buildings. Wind flow in this layer is entirely three-dimensional and difficult to illustrate by an empirical model. This is the core issue for simulating flow field and pollutant dispersion inside urban street canyons. To simplify investigation, we studied flat roofs with identical heights for two-dimensional street canyons. Based on the physical simplification, a morphometric method was brought into the model to represent different oncoming wind flows. The morphometric method can quantitatively represent aerodynamic parameters of upcoming wind flows. Field measured data of turbulence data was also used in the model to represent upcoming wind profiles. In Scenario Study A, we quantitatively investigated the effect of oncoming wind profiles and ARs on flow fields and pollutant characteristics inside street canyons.

Results showed that increasing oncoming wind speeds induced pollutant concentrations inside street canyons. Shear stress at roof level extended deeper into street canyons and enhanced flow exchange between external and internal flows. The morphometric method revealed that medium-sized buildings had the highest roughness length, which was good for the dispersing pollutant out of street canyons.

However, external wind flow cannot influence street-canyon flow regimes, which are dominated by ARs. The effect of three-dimensional wind flow at roof level on flow fields and pollutant dispersion inside street canyons needs further investigation, including more reliable wind speed and turbulence properties.

How does roof shape affect the flow field and pollutant dispersion inside street canyons?

The effect of roof shape on flow fields and pollutant dispersion inside street canyons had a demonstrated difference between rural and urban areas. In rural areas, upcoming wind profile is exactly ruled by logarithm law in a vertical direction. The turbulence profile in a vertical direction was modeled by Counihem (1975). When wind blows into street canyons, flow separation is seen in the leading edge of upwind buildings. This was shown in Scenario Study B. With different street-canyon configuration, different turbulence intensity distributions were seen inside and outside street canyons. Compared to even and step-up configurations, the step-down configuration had less ability to disperse pollutant. A high pollutant concentration was seen inside street canyons due to a second vortex generated at roof level. This prevented pollutant transportation out of street canyons. Among four roof shapes, the left-wedge roof had better ability to transport pollutant out of street canyons since strong flow separation happened at the leading edge of upwind buildings. Thus, high turbulence intensity at roof level of street canyons improves pollutant dispersion.

In urban areas, wind flow blows through inhomogeneous buildings. Roughness elements will generate an intensified shear layer at roof level due to flow separation. In the presence of buildings, zero wind speed is lifted to roof level. The generated shear layer is accompanied by chaotic vortices at roof level. Pollutant dispersion is then constrained by turbulence diffusion across the shear layer. As Kastner-Klein et al. (2004) noted, the chaotic
vortex hampered vortex formation inside street canyons. Flow inside street canyons became almost stagnant and trapped pollutant. The situation worsened for step-down building configuration. These inhomogeneous elements at upwind fetch are beneficial because they can produce high turbulent intensity for upwind profile.

*What are the weaknesses of present mathematical models of air quality prediction and the future work?*

Flow patterns and vehicle pollutant dispersion inside two-dimensional street canyons are systematically investigated in this thesis. A morphometric model, numerical turbulent $k-\varepsilon$ model and non-chemical reaction vehicle line pollutants were combined for mathematical modeling to predict airflow and pollutant concentration inside street canyons. The morphometric method was based on wind-tunnel experiments and a homogeneous assumption. Different morphometric methods demonstrated deviation predictions of aerodynamic parameters, especially for medium-spaced buildings. In real urban environments, wind speed and turbulence structure were significantly influenced by inhomogeneous of ground roughness elements. Variations in wind direction had a large influence on aerodynamic parameters. The uncertainty of wind speed and turbulence structure was enlarged by inhomogeneous elements and wind direction at the vicinity of roof level. In field measurements and wind-tunnel experiments, an intensified shear layer existed at roof level that cannot be illustrated by morphometric methods. However, the intensified shear layer strongly influenced pollutant dispersion inside street canyons. The morphometric method should be properly applied in densely spaced buildings to describe upcoming wind profiles.

Numerical turbulence $k-\varepsilon$ model was used in modeling, as well as default constant values of turbulence model, such as $C_\mu$ and turbulent Schmidt number $Sc_t$. However, grid distribution and turbulence model had large effect on distribution of pollutant concentrations inside street canyons. At the vicinity of roof level, a comparison between $k-\varepsilon$ and $k-\omega$ models revealed significant differences for predicting flow and pollutant concentrations inside street canyons. Therefore, advanced turbulence models must be further developed for reliable simulation. Reliably predicting flow field and pollutant distribution fields simultaneously is an interesting research topic. Currently, there are no convincing references on this topic to the author’s knowledge.

In the thesis, we suggested that at least 30 to 50 cells should be used to per building height. For pollutant dispersion simulation, 10 cells need to be put at ground level to capture high gradient concentration fields. In real urban circumstances, COST Franke et al., (2004) and AIJ guidelines (Tominaga et al., 2005) suggested that at least 10 cells should be used per building side. Further grid analysis is needed to study real industrial applications in three-dimension.

Since flow structure and pollutant concentrations inside street canyon are mainly dominated by local building geometry, further work will also focus on flow patterns and pollutant dispersion emitted by vehicles in a real complex three-dimensional environment, and validated by wind-tunnel experiments. Transportation of pollutants from external the environment into internal buildings will provide interesting future work. Solar-induced
thermal effects on flow and pollutant dispersion inside street canyons are also in the process of investigation.
References


