Doctoral Thesis

Time-resolved measurement and simulation of local scale turbulent urban flow

Author(s):
Immer, Marc Christian

Publication Date:
2016

Permanent Link:
https://doi.org/10.3929/ethz-a-010657618

Rights / License:
In Copyright - Non-Commercial Use Permitted
TIME-RESOLVED MEASUREMENT AND SIMULATION OF LOCAL SCALE TURBULENT URBAN FLOW

A thesis submitted to attain the degree of DOCTOR OF SCIENCES of ETH ZURICH (Dr. sc. ETH Zurich)

presented by MARC CHRISTIAN IMMER

Master of Science ETH in Mechanical Engineering
born March 26th, 1983
citizen of Oberhofen am Thunersee, BE

accepted on the recommendation of

Prof. Dr. Jan Carmeliet, examiner
Prof. Dr. GertJan van Heijst co-examiner
Prof. Dr. Patrick Jenny co-examiner
Dr. Jonas Allegrini, co-examiner

2016
ABSTRACT

With continuing urbanization and increasing population size, the understanding of the urban microclimate becomes more important. The urban microclimate is the specific climatic conditions found between and above buildings, meaning wind speed, air temperature, humidity and pollutant concentrations. These conditions can vary locally based on the urban morphology. The local microclimate has an influence on the building energy demand, pedestrian wind comfort and health of the inhabitants.

This research aimed to improve the understanding of the local scale urban flow field. Specifically, the influence of turbulence present in the flow above street canyons on the behaviour of shear layers at the street canyon top was studied. Due to the instationary nature of turbulent flows, time resolved techniques were employed. Wind tunnel measurements were conducted in the ETHZ/EMPA Atmospheric Boundary Layer (ABL) wind tunnel using a time resolved stereoscopic Particle Image Velocimetry (PIV) system and flow simulations were conducted on the High Performance Computing (HPC) facilities of EMPA and ETHZ using time resolved Large Eddy Simulation (LES).

The wind tunnel experiment simulated a large range of turbulent flow conditions for a unit aspect ratio street canyon geometry, by using a purpose-build wind tunnel setup. A split-floor setup and turbulence generating spires and barriers were used to vary the degree of turbulence. Time resolved stereo-PIV data was used for a detailed investigation of selected cavity flows. Spatio-temporal visualizations of vortex cores using the Q criterion were superimposed on sweep and ejections events. This provided useful insights on the dynamics of shear layers and the interaction with external turbulence. It was found that moderately turbulent cases feature shear layers that produce vortices through Kelvin-Helmholtz instabilities and cases with high turbulence showed vortex shedding at the upstream edge, caused by sweep events. The time dependent analysis of the highly turbulent case showed intermittent, large scale sweep events that penetrate into the cavity.

Flow simulations with LES were conducted to investigate the turbulent flow of a unit aspect ratio street canyon. For the inlet boundary con-
dition, artificial turbulence generation was used. A high quality of the simulations was achieved through a validation with the wind tunnel measurements. The validation showed that artificial turbulence generation is a viable approach. Additionally, the validation showed the need for high quality, time resolved stereo-PIV data. The simulations were used to investigate the removal of a passive scalar from the street canyon under different turbulent flow conditions. This revealed two distinct flushing mechanisms, transport through the shear layer and transport out of the cavity vortex. Turbulent inflow conditions significantly improved the rate of removal. Furthermore, a demonstration case for a full scale apartment building in an urban context showed the potential of LES in combination with a turbulent inflow generator to study local scale turbulent flow phenomena.

The work for this dissertation\(^1\) was performed at the Laboratory for Building Science and Technology at EMPA. I would like to express my thanks to the lab, and especially to Viktor Dorer, who led the Urban Physics Group for the majority of my stay. He supported me throughout my time at EMPA and always had an open door. The technical and administrative staff at EMPA also contributed substantially to my success. Beat Margelisch’s excellent work – from maintaining the wind tunnel to supporting the model’s fabrication and installation and generally his dedication and ingenuity in finding practical and rapid solutions – was invaluable. Additionally, Roger Vonbank and Stephan Carl’s support of the wind tunnel testing is greatly appreciated. The wind tunnel testing involved in this work was also supported by Felix Rubin and Michael Ammann from ZHAW, who lent me their hot wire calibration equipment. The flow simulations were performed on the HPC facilities at EMPA. I would like to thank the Ipazia cluster team, especially Patrik Burkhalter, for their exceptional technical support.

I would like to thank Dr. Peter Moonen for the introduction to the topic of this thesis and his scientific guidance as my direct supervisor for a large part of my dissertation. Working together on conference papers was an incredibly valuable educational experience for me. He was always available for a discussion and ready to share advice.

My thanks also go to me colleagues and co-workers – both for inspiring scientific discussions and a good work climate. My special thanks go to Marcel Vonlanthen. My work benefited greatly from our collaboration and discussions. His introducing me to the Python language changed my workflow completely and enabled me to perform rapid and accurate scientific analysis. Our open source projects are a testimony to our synergistic relationship. I would also like to thank Marcel for providing me with ABL precursor data for my LES simulations.

I would like to acknowledge the work that Stefan Ritz and Isabel Kendall performed as part of their Masters’ studies during my time

\(^1\) This research work was additionally supported by the Hartmann Müller-Fonds on grant ETH-32 11-2
at EMPA. Their work inspired me and also contributed to my thesis. Additional thanks go to Prof. Zheng-Tong Xie, who provided me with his implementation of the filtered noise inflow generator, and to Prof. Markus Klein, for the extensive discussions and advice on digital filtering.

I would like to thank my doctoral committee for critically reading my thesis and posing interesting and challenging questions. My thanks go to Prof. Dr. Jan Carmliet for the opportunity to perform this work in his group.

I would like to thank the Arbeitsgruppe für Luft- und Raumfahrt ALR, especially Dr. Georges Bridel, for the opportunity to continued to lead ALR’s Aircraft Performance Team during my PhD and the generous financial support during the writing period of my thesis.

The continuous support from friends and family over the last years made it possible for me to dedicate this time to my work and ultimately succeed. For this I am immensely grateful.
NOTATION

OPERATIONS

<table>
<thead>
<tr>
<th>SYMBOL</th>
<th>MEANING</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\bar{\cdot}$</td>
<td>temporal mean</td>
</tr>
<tr>
<td>$(\cdot)'$</td>
<td>mean subtracted value</td>
</tr>
<tr>
<td>$\tilde{\cdot}$</td>
<td>filtered value</td>
</tr>
<tr>
<td>$</td>
<td>\cdot</td>
</tr>
<tr>
<td>$\mathcal{F}$</td>
<td>Fourier transform</td>
</tr>
<tr>
<td>$\ast$</td>
<td>convolution</td>
</tr>
<tr>
<td>$\ast$</td>
<td>correlation</td>
</tr>
</tbody>
</table>

VELOCITY, FLOW AND FLUID

<table>
<thead>
<tr>
<th>SYMBOL</th>
<th>MEANING</th>
<th>UNIT</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u$</td>
<td>velocity</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>TKE / $\kappa$</td>
<td>turbulent kinetic energy</td>
<td>m$^2$ s$^{-2}$</td>
</tr>
<tr>
<td>$P_{TKE} / P_{\kappa}$</td>
<td>production of turbulent kinetic energy</td>
<td>m$^3$ s$^{-3}$</td>
</tr>
<tr>
<td>$l$</td>
<td>turbulence intensity</td>
<td>%</td>
</tr>
<tr>
<td>$\Phi$</td>
<td>power spectral density</td>
<td>m$^2$ s$^{-1}$</td>
</tr>
<tr>
<td>$f$</td>
<td>frequency</td>
<td>s</td>
</tr>
<tr>
<td>$\phi$</td>
<td>scalar concentration</td>
<td>-</td>
</tr>
<tr>
<td>$\nu$</td>
<td>kinematic viscosity</td>
<td>m$^2$ s$^{-1}$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>fluid density</td>
<td>kg m$^{-3}$</td>
</tr>
</tbody>
</table>
### NOTATION

### STATISTICS

<table>
<thead>
<tr>
<th>SYMBOL</th>
<th>MEANING</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E[\cdot]$</td>
<td>expected value</td>
</tr>
<tr>
<td>$\sigma/Std[\cdot]$</td>
<td>standard deviation</td>
</tr>
<tr>
<td>$Var[\cdot]$</td>
<td>variance</td>
</tr>
<tr>
<td>$Cov[\cdot]$</td>
<td>covariance</td>
</tr>
<tr>
<td>$\rho_i$</td>
<td>correlation coefficient</td>
</tr>
</tbody>
</table>

### INFLOW GENERATOR

<table>
<thead>
<tr>
<th>SYMBOL</th>
<th>MEANING</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon$</td>
<td>Gaussian random number</td>
</tr>
<tr>
<td>$\xi$</td>
<td>Gaussian random field</td>
</tr>
<tr>
<td>$v$</td>
<td>filtered field</td>
</tr>
</tbody>
</table>
# Acronyms

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABL</td>
<td>Atmospheric Boundary Layer</td>
</tr>
<tr>
<td>ACF</td>
<td>Autocorrelation Function</td>
</tr>
<tr>
<td>ARMA</td>
<td>Autoregressive Moving-Average</td>
</tr>
<tr>
<td>AR</td>
<td>Autoregressive</td>
</tr>
<tr>
<td>BC</td>
<td>Boundary Condition</td>
</tr>
<tr>
<td>BES</td>
<td>Building Energy Simulation</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CTA</td>
<td>Constant Temperature Anemometry</td>
</tr>
<tr>
<td>DEHS</td>
<td>Di-Ethyl-Hexyl-Sebacat</td>
</tr>
<tr>
<td>DMD</td>
<td>Dynamic Mode Decomposition</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>FFID</td>
<td>Fast Response Flame Ionization Detector</td>
</tr>
<tr>
<td>FFT</td>
<td>Fast Fourier Transform</td>
</tr>
<tr>
<td>FIR</td>
<td>Finite Impulse Response</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>HPC</td>
<td>High Performance Computing</td>
</tr>
<tr>
<td>HW</td>
<td>Hot Wire</td>
</tr>
<tr>
<td>LDA</td>
<td>Laser Doppler Anemometry</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LIF</td>
<td>Laser Induced Fluorescence</td>
</tr>
<tr>
<td>MA</td>
<td>Moving-Average</td>
</tr>
<tr>
<td>MPI</td>
<td>Message Passing Interface</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>PSD</td>
<td>Power Spectral Density</td>
</tr>
<tr>
<td>PTV</td>
<td>Particle Tracking Velocimetry</td>
</tr>
<tr>
<td>Acronym</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>----------------------------------</td>
</tr>
<tr>
<td>QLS</td>
<td>Quantitative Light Sheet</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>SE</td>
<td>Standard Error</td>
</tr>
<tr>
<td>TKE</td>
<td>Turbulent Kinetic Energy</td>
</tr>
<tr>
<td>TVD</td>
<td>Total Variation Diminishing</td>
</tr>
<tr>
<td>UAV</td>
<td>Unmanned Aerial Vehicle</td>
</tr>
<tr>
<td>UBL</td>
<td>Urban Boundary Layer</td>
</tr>
<tr>
<td>WSS</td>
<td>Wide-Sense Stationary</td>
</tr>
</tbody>
</table>
## CONTENTS

**NOTATION** ix  
**ACRONYMS** xi  

### 1 INTRODUCTION

1.1 Motivation ............................................ 1  
1.2 Scope and Objectives ................................. 3  
1.3 Outline ................................................. 4  

### 2 STATE OF THE ART

2.1 Atmospheric Scales .................................... 5  
2.2 Atmospheric Boundary Layer .......................... 6  
2.3 Turbulence ............................................. 7  
2.4 Local Scale Urban Flows ............................... 9  
2.5 Cavity and Street Canyon Flow ....................... 12  
2.6 Experimental Methods ................................. 14  
  2.6.1 Boundary Layer Modeling ......................... 14  
  2.6.2 Measurement Equipment ........................... 15  
  2.6.3 Flow Simulations ................................ 19  
  2.6.4 Turbulent Inlet Boundary Condition .............. 21  
2.7 Summary and Need for Further Research ............. 24  

### 3 BACKGROUND

3.1 Governing Equations ................................... 27  
3.2 Statistical Description of Turbulence ................ 28  
  3.2.1 First and Second Order Moments .................. 29  
  3.2.2 RANS Equations ................................ 30  
  3.2.3 Convolution and Spectra .......................... 32  
  3.2.4 Correlation and Integral Scale ..................... 33  
3.3 Time Series Analysis ................................. 36  
3.4 Flow Diagnostic ...................................... 38  
  3.4.1 Quadrant Analysis ................................. 38  
  3.4.2 Vortex Identification ............................. 38  
3.5 Conclusion ............................................ 39  

### 4 CAVITY FLOW EXPERIMENT

4.1 Experimental Setup ..................................... 42  
  4.1.1 Experimental Facility .............................. 42
CONTENTS

4.1.2 Measurement System ................................. 43
4.1.3 Model Geometry ................................. 44
4.1.4 PIV Vector Processing and Velocity Post-Processing 46
4.2 Cases and Measurements ................................. 47
4.2.1 PIV Flow Measurements ................................. 48
4.3 Uncertainty Estimation ................................. 48
4.3.1 Experiment Design ................................. 49
4.3.2 Measurement Uncertainty ................................. 50
4.3.3 Statistical Uncertainty ................................. 50
4.3.4 Uncertainty estimation of the integral time scale ........ 52
4.4 Spire Flow Field ................................. 55
4.4.1 Hot Wire Measurements ................................. 56
4.4.2 Results ................................. 56
4.5 Cavity Vortex ................................. 62
4.6 Shear Layer Analysis ................................. 66
4.6.1 Inlet Flow and Cavity Flow Field ................. 66
4.6.2 Shear Layer ................................. 70
4.6.3 Discussion ................................. 73
4.7 Discussion and Conclusion ................................. 87
5 COMPUTATIONAL MODEL FOR THE URBAN AIR 89
5.1 Scope ................................. 89
5.2 Scale and Computational Domain ................................. 90
5.3 Large Eddy Simulation (LES) ......................... 90
5.3.1 Filtered Navier-Stokes Equations ................. 90
5.3.2 SGS Model ................................. 91
5.3.3 Scalar Transport Model ................................. 92
5.3.4 Discretization ................................. 93
5.3.5 Computational Grid ................................. 93
5.3.6 Boundary Conditions ................................. 93
5.3.7 Initial Condition & Convergence ................. 94
5.3.8 Averaging and Runtime Sampling ................. 97
5.4 Implementation ................................. 98
5.5 Conclusion ................................. 98
6 TURBULENT INFLOW GENERATION 99
6.1 Evaluation of Methods ................................. 99
6.2 Filtered Noise Method ................................. 104
6.2.1 Procedure ................................. 104
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.2.2</td>
<td>Simplifications</td>
<td>107</td>
</tr>
<tr>
<td>6.3</td>
<td>Present Method</td>
<td>107</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Spatial Correlation</td>
<td>108</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Temporal Correlation</td>
<td>111</td>
</tr>
<tr>
<td>6.3.3</td>
<td>Implementation</td>
<td>112</td>
</tr>
<tr>
<td>6.3.4</td>
<td>Discussion</td>
<td>114</td>
</tr>
<tr>
<td>6.4</td>
<td>Application to Boundary Layer</td>
<td>116</td>
</tr>
<tr>
<td>6.4.1</td>
<td>Computational Setup</td>
<td>117</td>
</tr>
<tr>
<td>6.4.2</td>
<td>Inlet Data</td>
<td>118</td>
</tr>
<tr>
<td>6.4.3</td>
<td>Cases</td>
<td>119</td>
</tr>
<tr>
<td>6.4.4</td>
<td>Results</td>
<td>121</td>
</tr>
<tr>
<td>6.5</td>
<td>Turbulent Scalar</td>
<td>136</td>
</tr>
<tr>
<td>6.5.1</td>
<td>Method</td>
<td>137</td>
</tr>
<tr>
<td>6.5.2</td>
<td>Computational Setup</td>
<td>139</td>
</tr>
<tr>
<td>6.5.3</td>
<td>Results</td>
<td>139</td>
</tr>
<tr>
<td>6.6</td>
<td>Discussion and Summary</td>
<td>141</td>
</tr>
<tr>
<td>7</td>
<td>Application of the Urban Air Model</td>
<td>145</td>
</tr>
<tr>
<td>7.1</td>
<td>Cavity Flow Validation</td>
<td>146</td>
</tr>
<tr>
<td>7.1.1</td>
<td>Computational Setup</td>
<td>146</td>
</tr>
<tr>
<td>7.1.2</td>
<td>Domain and Cases</td>
<td>146</td>
</tr>
<tr>
<td>7.1.3</td>
<td>Results</td>
<td>147</td>
</tr>
<tr>
<td>7.1.4</td>
<td>Conclusion</td>
<td>153</td>
</tr>
<tr>
<td>7.2</td>
<td>Turbulent Street Canyon</td>
<td>154</td>
</tr>
<tr>
<td>7.2.1</td>
<td>Computational Setup</td>
<td>154</td>
</tr>
<tr>
<td>7.2.2</td>
<td>Flow Fields</td>
<td>155</td>
</tr>
<tr>
<td>7.2.3</td>
<td>Flushing</td>
<td>160</td>
</tr>
<tr>
<td>7.2.4</td>
<td>Conclusion</td>
<td>164</td>
</tr>
<tr>
<td>7.3</td>
<td>Case Study</td>
<td>166</td>
</tr>
<tr>
<td>7.3.1</td>
<td>Inlet Profiles</td>
<td>167</td>
</tr>
<tr>
<td>7.3.2</td>
<td>Computational Setup</td>
<td>168</td>
</tr>
<tr>
<td>7.3.3</td>
<td>Results</td>
<td>170</td>
</tr>
<tr>
<td>7.3.4</td>
<td>Discussion and Summary</td>
<td>181</td>
</tr>
<tr>
<td>7.4</td>
<td>Conclusion</td>
<td>182</td>
</tr>
<tr>
<td>8</td>
<td>Discussion and Conclusion</td>
<td>185</td>
</tr>
<tr>
<td>8.1</td>
<td>Summary</td>
<td>185</td>
</tr>
<tr>
<td>8.2</td>
<td>Contributions to the research field</td>
<td>187</td>
</tr>
<tr>
<td>8.3</td>
<td>Outlook and Future Work</td>
<td>188</td>
</tr>
</tbody>
</table>
CONTENTS

BIBLIOGRAPHY 191
INTRODUCTION

1.1 MOTIVATION

With the world’s growing population, currently over seven billion people, there is a continuous trend towards urbanization. The United Nations report on the prospects of urbanization (United Nations 2014) states that in 2014, more than half (54%) of the world’s population lived in urban area. By 2050, this number is projected to increase to 66%. Interestingly, urbanization does not just concern to mega-cities. Half of the urban population lives in settlements with less than 500,000 people. The report concludes with the following statement: “As the world continues to urbanize, sustainable development challenges will be increasingly concentrated in cities.”

With this continuing urbanization and increasing population size, the understanding of the urban microclimate becomes more important. The urban microclimate is the specific climatic conditions found between and above buildings, meaning wind speed, air temperature, humidity and pollutant concentrations. Based on the urban morphology (the size, shape and arrangement of buildings and vegetation) these conditions can vary locally. Heat, humidity and pollutants are transported by local turbulent flows in and out of the street level and mixed with the Atmospheric Boundary Layer (ABL) flow aloft.

The urban microclimate has an influence on the building energy demand, in particular the space heating and cooling demand (Allegrini 2012). Building Energy Simulation (BES) is used to calculate the energy demand of new and existing buildings by the means of an annual time resolved simulation. BES accounts for the energy exchange between the building envelope and the environment by using simplified assumptions regarding air temperature, mean radiant temperature and wind speed. Climate data from a nearby weather station such as an airport is often used. The validity of this assumption is questionable for buildings in an urban context due to the locally different conditions. This was found to contribute to the difference of the predicted versus actual for the energy consumption of a building, the so called performance gap. Sun et al. (2014)
states with regard to the performance gap that "given the simplicity of the current standard micro climate calculations, it is expected that the uncertainty stemming from using these calculations has significant influence on the uncertainty in the simulated energy outcomes." Better prediction of the urban microclimate therefore has the potential to improve BES models by enabling a more efficient design of heating and cooling systems (active and passive) and therefore ultimately resulting in a reduction of the energy demand of new and retrofitted buildings.

Climatic and mesoscale weather models do not have sufficient spatial resolution to capture individual buildings, therefore urban areas are modelled using a parameterization of the exchange fluxes of momentum, heat and air composition from the buildings to the ABL flow. Such parameterizations often use the concept of urban canyons (Oleson et al. 2008). A better understanding of the urban wind field can lead to better parameterizations, therefore improve weather predictions and help to better understand the role of urbanization in climate predictions.

The urban wind flow field also impacts comfort and health (Saneinejad 2013). The interaction of building flow fields can create areas with locally high wind speeds and variability of wind direction, impacting pedestrian wind comfort. Additionally, low wind speed areas in street canyons and courtyards can trap pollutants and heat, causing health problems.

Additionally, a better understanding of the urban flow field also benefits other fields, such as the safe operation of Unmanned Aerial Vehicles (UAVs) in urban areas (Galway et al. 2008).

The various aspects of urban microclimate and its effects on the population was studied by previous researchers, i.e. described in the review paper on Urban Physics by Moonen et al. (2012b). Significant advances in the prediction of the urban wind flow field by Computational Fluid Dynamics (CFD) methods was achieved in the field of Computational Wind Engineering, e.g. described in the review papers by Blocken (2014) and Blocken (2015). The progress of the modelling of the ABL above urban areas, the so called Urban Boundary Layer (UBL), is described in the review paper by Barlow (2014).

Most computational studies use Reynolds-Averaged Navier-Stokes (RANS) simulations to predict flow field statistics such as mean wind velocity. Turbulence is not simulated but parameterized. As flows in urban environments are highly turbulent and instationary and turbulence plays an important part in the transport and mixing of heat, humidity and
pollutants from the streets level to the flow aloft, such parameterizations might not always be justified. To improve these models, there is a need for high quality time resolved simulations such as Large Eddy Simulation (LES). LES based studies are not as widely used as RANS, due to the high computational cost. The computational power currently available does allow for LES to be applied for problems of increasing complexity (Pope 2004). This creates the need for high quality, high resolution time dependent validation data. The advances in wind tunnel measurement equipment, especially in the area of imaging techniques such as Particle Image Velocimetry (PIV), can provide such data.

1.2 Scope and objectives

The aim of this research is to improve the understanding of the local scale urban flow field. Specifically, the influence of turbulence present in the flow above street canyons on the behaviour of shear layers at the street canyon top will be studied. Due to the instationary nature of turbulent flows, time resolved techniques will be employed.

Wind tunnel measurements will be conducted in the ETHZ/EMPA Atmospheric Boundary Layer (ABL) wind tunnel using a time resolved stereoscopic PIV system and flow simulations will be conducted on the High Performance Computing (HPC) facilities of EMPA and ETHZ using time resolved LES.

The wind tunnel experiment aims at simulating a large range of turbulent flow conditions for a street canyon geometry by using a purpose-build wind tunnel setup. Time resolved stereo-PIV data will be used for a detailed investigation of selected cavity flows. Additionally, high quality datasets for LES validation will be created.

LES will be conducted to investigate the turbulent flow of a street canyon. A high quality of the simulations will be achieved by a validation with the wind tunnel measurements. The removal of a passive scalar from a street canyon under turbulent flow conditions will be investigated. Furthermore, the state of the art for urban LES will be improved by studying artificial turbulence generation methods.
1.3 Outline

This thesis consists of eight chapters. Chapter 1 provides an introduction to the field, states the aim and objectives of the thesis and provides an outline.

Chapter 2 provides a state of the art of research on urban flow fields. Based on this, the need for further research is described.

Chapter 3 describes the governing equations and the statistical and mathematical tools used to describe turbulent flows.

Chapter 4 contains the main experimental part of this thesis. The wind tunnel facility and measurement PIV and Constant Temperature Anemometry (CTA) measurement equipment is described. An estimate on the uncertainty of the PIV measurements is provided and a new method to estimate uncertainty for integral scales is described. The measurements for cavity flows under different turbulent conditions are presented and specific cases are investigated and discussed in detail.

Chapters 5, 6 and 7 contain the numerical part of the thesis. Chapter 5 describes the model used to simulate the urban air flows. The model uses the LES approach to simulate turbulence.

Chapter 6 describes the synthetic turbulence generator used as the inlet boundary condition in the urban air model. The method is described in detail and a formal way to obtain the inflow generator parameters is derived. Simulations show the effect of different parameter on the produced flow field. Specific details with regard to the code implementation are given.

Chapter 7 applies the computational model to three specific cases of increased complexity. The influence of different grid resolutions on a cavity flow is studied and compared with wind tunnel measurements. The capability of the inflow generator to provide the same conditions as were measured in the wind tunnel is investigated using the cavity geometry. Finally, the model is applied to an example of an urban geometry.

Chapter 8 concludes the thesis by providing a critical discussion, a summary and an outlook.
STATE OF THE ART

In this thesis time resolved wind tunnel measurements as well as time resolved simulations are used to investigate turbulent urban flows. This chapter provides a brief introduction to atmospheric and turbulent flows. The flow features encountered in urban flows are described, with a particular focus on the street canyon type flow. A description of different measurement techniques and simulation methods is provided.

2.1 ATMOSPHERIC SCALES

The Earth’s atmosphere is a layer of gases surrounding the planet, captured by its gravitational pull. The atmosphere extends as far as 500 – 1000 km above the surface, however 80% of the total mass is contained within a thin layer approximately 11 km thick (Wallace and Hobbs 2006). This lowest layer is called the troposphere. The troposphere is characterized by a steady decrease of temperature with altitude. The top of the troposphere is called the tropopause, defined by the altitude above which the temperature starts to rise again. The altitude of the tropopause is around 9 km at the poles and up to 17 km at the equator. The air in the troposphere is heated predominantly by the ground through convection. The ground absorbs around 47% of the total direct solar irradiance, compared to only 25% absorbed by the atmosphere (the remaining 28% is reflected back into space) (Oke 1987). The warm air from the ground is transported upwards through turbulent mixing. The temperature inversion at the tropopause acts as a barrier, limiting further upwards transport. Most of the atmospheric water vapor is therefore contained in the troposphere, meaning that this is where weather phenomena such as clouds and precipitation occur. More solar energy is received at the equator than at the poles, leading to a global circulation of air. Together with the influence of the earth’s rotation through the Coriolis force, characteristic large scale weather phenomena emerge such as cyclones and fronts. The time and length scale of these, compared to other major meteorological phenom-
Meteorological time and length scales of atmospheric flow phenomena, adapted after Schlünzen et al. (2011).

The global circulation changes over the duration of months and years (seasonal and climatic changes), whereas fronts and cyclones have timescales of multiple days to weeks. Smaller scale, regional phenomena such as seas breezes, valley winds and the urban heat island effect depend on the topography (terrain, vegetation, buildings, etc.) and the diurnal cycle. Cloud formation, thunderstorms and convection cells are typical weather phenomena that happen on a shorter time scale (hours).

2.2 ATMOSPHERIC BOUNDARY LAYER

Only the lowest part of the troposphere is directly affected by the planetary surface. This part is called the Atmospheric Boundary Layer (ABL) (Stull 1988). Above the ABL the wind flow is largely geostrophic, meaning it is mainly influenced by the pressure gradient and the Coriolis force. Two important phenomena that originate in the ABL are thermals and wake flows. The scales are shown in Figure 2.1 within the box denoted Local Scale. Thermals are pockets of air that are warmed up by the surface
and then lifted upwards through buoyancy. Wake flows are caused by surface obstacles that locally disturb the flow near the ground, such as terrain features and buildings. The flow is unable to follow the surface and detaches from it, causing reversed flow and therefore shear layers. Both thermals and wake flows are accompanied with the generation of vortices and turbulence.

2.3 TURBULENCE

Turbulence is observable as a seemingly random motion that occurs for flows of different scales and different fluids, from interstellar gas clouds to water in a river. Turbulence is of chaotic nature and almost always three dimensional. This makes it difficult to describe deterministically.

Turbulence develops through instability mechanisms, exited by small perturbations or inflection points in the velocity profile. Turbulence can only develop if the ratio of the inertial forces to the viscous forces is sufficiently high. This ratio is expressed by the Reynolds number:

\[ Re = \frac{uL}{\nu} \]  

(2.1)

where \( u \) is a characteristic velocity of the flow, \( L \) a characteristic length scale and \( \nu \) the kinematic viscosity of the fluid. The Reynolds number is named after Osborne Reynolds. In his famous experiment (Reynolds 1883), he showed that pipe flows get turbulent above a certain critical Reynolds number. Atmospheric flows and urban flows are almost always turbulent, due to the high Reynolds number.

The understanding of turbulence was significantly influenced by the work of Lewis Fry Richardson, who described turbulence as a collection of differently sized eddies undergoing an energy cascade (Richardson 1922) and by the contributions of Andrey Kolmogorov with regards to the properties of small scales eddies for large Reynolds numbers (Kolmogorov 1941). The energy cascade is visualized in Figure 2.2 as a transfer of energy from the large scales to the small scales. Turbulent kinetic energy is produced through a production mechanism at large scales. The largest eddies are of scale \( \ell_0 \) and are usually similar in size to the characteristic scale \( L \). This region is called the energy containing range. The larger eddies break up into smaller eddies by inviscid processes, transferring their energy to successively smaller scales. According to Kolmogorov’s hy-
Figure 2.2: Turbulence energy cascade and spectrum, adapted after Pope (2000).

The hypothesis of local isotropy, eddies smaller than $\ell_{EI}$ are statistically isotropic. This means they lost the information about the directionality of the large scales through the cascade process. The smallest eddies are of the size $\eta$, the Kolmogorov length. After that, they are dissipated into heat by the viscosity of the fluid. Turbulence can therefore not sustain itself without a production mechanism, as all turbulent motions would dissipate and the flow would return to a laminar state.

Turbulence is not just a random motion, but composed of organized flow structures. The concept of eddies can therefore by further extended to the concept of coherent structures. A definition of a coherent structure is proposed by Robinson (1991b): a three-dimensional region of the flow over which at least one fundamental flow variable (velocity component, density, temperature, etc.) exhibits significant correlation with itself or with another variable over a range of space and/or time that is significantly larger than the
Figure 2.3: Model for coherent boundary structures, from Robinson (1991a). Momentum is exchanged by streamwise vortices near the wall and horseshoe (or hairpin) like vortices through sweep and ejection.

The smallest local scales of the flow. Coherent structures therefore are not limited to just vortices but more generally describe correlated motion. A concept for boundary layer coherent structures is presented in Figure 2.3. Vortical structures in the shape of streamwise vortices and horseshoe/hairpin vortices result in a momentum exchange through sweep and ejection events with the high/low speed flow. The streamwise vortices were observed in boundary layer visualizations by Kline et al. (1967) as streak-like patterns. More on the detection of coherent structures and sweep/ejection events is given in Chapter 3.4.

2.4 LOCAL SCALE URBAN FLOWS

In Figure 2.1, turbulence was placed at the small scales, even tough turbulence phenomena are present at all scales. At the local scale, turbulence dominates the flow. It is responsible for the mixing and transport of heat, moisture and pollutants. When approaching an urban area, the ABL adapts to the new surface roughness by developing the so called
Urban Boundary Layer (UBL). A detailed description of the UBL is given by Barlow (2014). At the lower levels, the UBL is characterized by flow patterns that develop around buildings. Figure 2.4 shows some of the occurring flow features. The flow approaching a building will separate into a stream pushed above the building and a stream deflected downwards. The downward flow rolls up into a characteristic horseshoe vortex, whereas the upwards flow detaches at the upstream roof edge of the building. A separation bubble forms that may re-attach on the roof. Flow separation also occurs at the downstream roof edge, creating a wake flow. These flow features are three dimensional, as depicted in Figure 2.5 for the flow around a simple cube immersed in a boundary layer. The horseshoe vortex can be seen wrapping around the building (hence the name) and flow separation is observed both on the top and the sides of the cube. Behind the cube, a large flow separation zone (wake) is visible. The flow around a cube represents a highly simplified case. Real buildings are geometrically much more complex and are rarely aligned with the flow direction. The flow field around complex shapes is however still composed of the above mentioned flow phenomena. For low wind speeds, buoyancy effects become dominant (through solar heating of surfaces) and significantly different flow fields are observed.

In urban areas, additional complexity to the flow field is added by the interaction between buildings. The wake flow of a building might interact with a downstream building. Based on wind tunnel measurements conducted by Hussain and Lee (1980), Oke (1988) categorizes three types of interactions, as depicted in Figure 2.6. Isolated roughness flow occurs, when the flow separation zone behind a building re-attaches to the ground before encountering another building. This situation may exists in rural or suburban areas that are lightly built up, but can also exist in urban areas where a large empty space such as a park, lake or river separates buildings. If buildings are closer together, the wake directly interacts with the downstream building. This is called wake interference flow. This results in a strong mixing of the wake flow and high momentum flow of the boundary layer. For buildings that are separated only by a small gap, such as a street, a typical wake flow does not develop, the flow directly re-attaches at the downstream building roof edge. Since the flow above the buildings therefore is not directly affected by the buildings geometry, this situation is called skimming flow. A lower velocity vortex (or vortex
Figure 2.4: Urban Boundary Layer (UBL) developing over a city (adapted after Oke 1987) and flow phenomena occurring at a local scale.

Figure 2.5: Flow topology of a cube immersed in a turbulent channel flow, from Martinuzzi and Tropea (1993).
system) exists between the buildings. This vortex is shown in Figure 2.4 denoted street canyon vortex.

2.5 CAVITY AND STREET CANYON FLOW

The skimming flow regime (Figure 2.6) occurs for the so called street canyon geometry. The street canyon is a heavily simplified model for an urban geometry and is defined as the space in between two elongated buildings. Under perfectly perpendicular flow conditions, the street canyon top is the only way the street level can exchange air, and therefore pollutant and heat, with the flow aloft (Figure 2.7).

Due to its geometrical simplicity, the street canyon was used to create models for pollutant dispersion (e.g. Dabberdt et al. 1973; Johnson et al. 1973; Nicholson 1975) and thermal models (e.g. Nunez and Oke 1977). The apparent simplicity of the street canyon is contrasted by the complexity of the flow field: the low velocity flow in the canyon and the high velocity of the flow over the buildings form a distinct shear layer at the canyon top. This shear layer is at the interface of the canyon air and the air aloft. The shear layer was observed in field measurements and is turbulent and unsteady (Louka et al. 2000). With a turbulent upstream flow, such as generated by an UBL, the cavity shear layer shows properties of a free
shear layer, such as vortex formation through Kelvin-Helmholtz type instabilities (Cui et al. 2004; Salizzoni et al. 2011).

The street canyon geometry is more generally described as an open cavity. Under perpendicular incident flow, they are often also called shear-driven cavities. In a shear-driven cavity the interface is defined by a mixing layer between the cavity flow and the external boundary layer flow, in contrast to the lid-driven cavity, where the top is a closed moving surface. This shear layer is governed through complex dynamical phenomena, arising through flow instabilities that are influenced by the external flow and through feedback mechanisms between the impinging flow on the downstream edge, the cavity flow and the shear layer flow.

Shear-driven cavities have been initially researched for their relevance in aeronautics, where drag and vibrations were the primary interest, for example in panel gaps, wheel wells or weapon bays. The flow conditions for these studies are usually thin laminar or turbulent boundary layers at high Reynolds Numbers. The experimental wind tunnel research by Roshko (1955), Rossiter and Britain (1964) and Sarohia (1975) have explored the existence of resonance in the shear layer, generated through acoustic and hydrodynamic feedback. This phenomena is called self-sustained oscillations. It occurs when vortices in the cavity shear layer impinge on the downstream cavity edge, sending a pressure wave upstream that perturbs the unstable shear layer and causes vortex formation. This leads to a distinct vortex shedding frequency, subject to the cavity geometry and the upstream boundary layer (Rockwell and Naudascher 1979).

Figure 2.7: Street canyon geometry and flow field, showing a large main vortex and smaller street level corner vortices. From Dabberdt et al. (1973).
Both wind tunnel experiments measurements and simulations using Large Eddy Simulation (LES) have made significant contributions to the understanding of cavity type shear flows. Specifically, imaging measurement techniques such as Particle Image Velocimetry (PIV) (as conducted by e.g. Ashcroft and Zhang 2005; Kang et al. 2008; Haigermoser et al. 2008) in combination with mathematical analysis tools such as modal analysis (e.g. Basley et al. 2010) or POD (e.g. Kang and Sung 2009; Kellnerová et al. 2012) showed the complexity of the shear layer. Coherent structures form in the shear layer based on the influence and interaction with turbulent coherent structures present in the upstream boundary layer. It might therefore not be surprising that the detailed wind tunnel study conducted by Salizzoni et al. (2011) on a row of street canyons, did not show that the statistical properties of either the shear layer or the canyon flow can be scaled by a single velocity scale. The LES study by Chang et al. (2006) compared the flushing of a scalar from a cavity subjected to a laminar and a turbulent boundary layer and showed that the scalar removal takes place through unsteady coherent structures in the shear layer that additionally showed intermittent behaviour for the turbulent case.

2.6 EXPERIMENTAL METHODS

The urban flow field can be investigated experimentally in a wind- or water tunnel. The experiment should feature a turbulent boundary layer and the measurement equipment employed should be able to measure turbulent flows that are three dimensional and in the case of wake flows, not have a preferred direction.

2.6.1 Boundary Layer Modeling

Experimental modeling of urban flow fields can be conducted in a boundary layer wind tunnel. Boundary layer wind tunnels are equipped with an inlet fetch in front of the test section. This fetch is used to develop an appropriate turbulent boundary layer. The boundary layer should represent the roughness of the upstream conditions (e.g. buildings, vegetation) and correspond to the scale of the model. Boundary layer modeling is an essential part in wind engineering and norms have been developed (e.g.
A turbulent boundary layer can be naturally developed in a long upstream section modelling the terrain and buildings. As this requires a very long wind tunnel, artificial ways to generate a thick boundary layer using turbulence generating obstacles such as spires and barriers were developed (Armitt and Counihan 1968; Counihan 1969). The method was further developed and characterized by Irwin (1981) using triangular shaped spires. Further development on the Counihan spires was conducted by Kozmar (2009) and the size of the development section was further reduced by simulating only the lower part of the ABL (Kozmar 2011). Turbulent length scales of artificially thickened boundary layers are investigated by Varshney and Poddar (2011).

An alternative approach was proposed by Makita (1991) to use a grid of active flaps to generate turbulence. This method was used in combination with vertical strakes and floor roughness by Cal et al. (2010) to investigate wind turbines.

Recent studies, such as the measurements conducted by Hancock and Pascheke (2014) and Hohman et al. (2015) on barrier and spire type turbulence generators, show that wind tunnel boundary layer modeling is still an active field of research.

2.6.2 Measurement Equipment

Flow velocity measurements in wind tunnels can be conducted by intrusive and non-intrusive means. Intrusive measurements typically include probes, non-intrusive measurements include laser based measurements.
A Pitot probe measures the dynamic pressure by the difference between the static and the total pressure. From the dynamic pressure (and known fluid density) the velocity can be obtained. Pitot probes work best for higher velocities, as large pressure differences can be measured more precisely (Barlow et al. 1999). For low velocities, Constant Temperature Anemometry (CTA) can be used. A Hot Wire (HW) probe consists of a small wire that is heated by an electric current. Turbulent flows change the temperature of the wire by convective cooling and therefore change the electrical resistance of the wire. High speed electronics control the current required to keep the wire at a constant temperature. Through the law for convective cooling of a cylinder (called King’s Law), the control voltage can be related to the flow velocity. A typical CTA setup is depicted in Figure 2.9, showing the HW probe, the CTA control electronics and the data processing. Typical miniature wind tunnel probes have a wire diameter of 5 \( \mu m \). Due to the low thermal mass, HW probes react very fast to velocity changes, thus enabling turbulence measurement with a very high temporal resolution (Bruun 1995). HW probes have between one and three wires, depending on how many velocity components can be measured. The incident flow angle is limited, making hot wire probes not suitable for detached flows.

Laser Doppler Anemometry (LDA) can be considered a non-intrusive technique. The method is based around the reflection of laser light from particles present in the flow. Two laser beams form an intersecting volume featuring a fringe pattern. Laser light gets scattered by a particle crossing the measurement volume and from the Doppler shift of the scattered light the velocity of the particle can be computed. Multiple velocity
components can be measured using multiple laser beams of different colors (frequencies). For larger wind tunnels, due to the limited focal length between the optics and the intersection point, the LDA device is often housed in an aerodynamic fairing inside the wind tunnel, making it a semi-intrusive measurement.

**PIV** is a class of non-intrusive, imaging based measurement techniques. Particles are seeded into the flow and illuminated on a plane using a laser sheet. Figure 2.10 shows a typical **PIV** setup, consisting of a laser with sheet optics and a camera. Two images in rapid succession are taken using a high speed camera or a double-frame camera. During the image exposure, the laser is pulsed to illuminate the particles. This process is presented in Figure 2.11 for a double frame camera. The flow velocity can be determined from the particle displacement from one image to the other and the time interval between the images. The time between two successive images is denoted $dt_{PIV}$. This time is chosen to be small to not loose track of particles, but large enough to show a clear particle shift. The time between two image pairs ($dt_{acq}$) is related to the acquisition frequency of the measurement as $f_{acq} = 1/dt_{acq}$. The double frame technique (Fig. 2.11) is ideal for time averaged measurements as $dt_{PIV}$ can be chosen independently from $dt_{acq}$. For time resolved **PIV** and single frame images,
the maximum acquisition frequency (besides the hardware limitations) cannot be higher than $1/d_{PIV}$.

A common PIV method is cross-correlation PIV. This method computes the spatial correlation between the two particle images by shifting an interrogation window. The location of the maximum of the correlation peak will give the average particle displacement within the interrogation window. In contrast to Particle Tracking Velocimetry (PTV), where individual particles are tracked, cross-correlation PIV is robust with regards to the seeding density and the image (illumination) quality. Using one camera will give a 2D field of 2D flow vectors ($2D2C$ PIV or planar PIV). Using two cameras in stereoscopic arrangement, the velocity component normal to the image plane can be computed ($2D3C$ PIV or stereo PIV). Using multiple cameras and a thick laser sheet or volume, the velocity field in a volume can be measured. This is called volumetric or tomographic PIV. Due to the high laser power requirements, this technique is applicable only for small volumes. A detailed description of PIV is found in Raffel et al. (2007).

The measurement of transported scalar quantities can also be performed. Dispersion studies are typically conducted in a wind tunnel by measuring the concentration of a tracer gas using a Fast Response Flame Ionization Detector (FFID) probe.

Additional techniques to support measurements include flow visualization by smoke (wind tunnels) or by dye (water tunnels).
2.6.3 Flow Simulations

Computational Fluid Dynamics (CFD) can be used to study urban flows. The existing CFD methods can be separated by their respective treatment of turbulence.

Direct Numerical Simulation (DNS) solves the Navier-Stokes system of equations (conservation of mass and momentum) in a time depended way. All the non-linear mechanisms of turbulence production and the viscous dissipation are directly resolved. This requires a very fine computational grid. The main issue with DNS is the associated computational cost that grows with the Reynolds number as $Re^3$ (Pope 2000), making DNS impossible for high Reynolds number urban flows. DNS can still be employed to improve the understanding of urban flows, such as the simulations performed by Coceal et al. (2006) for an array of cubes. As the simulation was conducted at a lower Reynolds number than full scale, this can be regarded as a scale model simulation. This requires a careful evaluation if the results are influenced by the Reynolds number.

In contrast to the DNS, a Reynolds-Averaged Navier-Stokes (RANS) simulation does not resolve any scales of the turbulent spectrum. RANS solves for the steady solution for the mean flow field. The production, transport and dissipation of turbulent energy is modelled. The main advantage of RANS is its computational efficiency. RANS can be successfully applied for steady state problems where the boundary layers are attached. For detached flows over bluff bodies, the prediction by RANS simulations can be very sensitive to the selected turbulence model or the model coefficients (Gorlé et al. 2015). Additional shortcomings of RANS lie in prediction of flows with buoyancy (Tomas et al. 2015) and dispersion (Moonen et al. 2011; Tominaga and Stathopoulos 2011; Salim et al. 2011). The quality of RANS results however can be improved by following one of the numerous best practice guidelines that have been developed for the respective field of application. For urban flows these are Franke et al. (2007), Tominaga et al. (2008) and Blocken (2015). A RANS simulations can also be conducted unsteady (URANS), to account for slowly changing boundary conditions. This approach is only valid if the timescale of the changing boundary conditions is significantly longer than the timescales of turbulence.

Large Eddy Simulation (LES) can be regarded as a solution to the high computational cost of DNS. Based on Kolmogorov’s theory that turbulence is isotropic when the scales are sufficiently small, the turbulence spectrum
is cut into a large eddy part and a subscale part. The large eddies are directly simulated on a computational grid, whereas the small scales are modeled. This subgrid scale model replaces part of the energy transfer to smaller scales and the dissipation into heat (Fig. 2.2). LES was originally proposed by Smagorinsky (1963) for weather prediction and was further explored by (Deardorff 1970b) for ABL and channel flows (Deardorff 1970a).

The LES approach is applicable for a wide range of flows. LES is especially suited for problems with free shear layers, separated flows and geometries with sharp corners (Piomelli 1999). This was demonstrated e.g. by the wind tunnel comparison of the flow around a cube immersed in a turbulent boundary layer (Lim et al. 2009). Piomelli (1999) also states high Reynolds number wall bounded flows as one of the major problems of LES still to solve. Near the wall, the flow is dominated by the small scales and the LES approach is therefore questionable (Pope 2004; Kawai 2010). For problems dominated by the large scales, such as pollutant dispersion, LES was successfully applied for local scales (e.g. Gousseau et al. 2012; Moonen et al. 2013) and for scales as large as entire cities (Nozu and Tamura 2012; Nozu et al. 2015).

The quality of a LES is not only influenced by the choice of the subgrid model, but to a large extent by the mesh resolution. The assessment of the required mesh resolution was studied by multiple authors (e.g. Addad et al. 2008; Davidson 2009; Bazdidi-Tehrani et al. 2013). The difficulties of creating a clear grid resolution requirement is stated by Gant (2009) to be one of the major problems for the reliability of LES results. Grid independency studies as conducted for RANS simulations are not directly applicable to LES for two reasons. First, LES is computationally demanding and a reduction in the cell size by half results in a roughly sixteen times increase in computational time (due to eight times more cells and a half the timestep). Secondly, an increase of the mesh resolution will by definition of the LES method result in more information, due to resolving more of the smaller scales. Turbulent Kinetic Energy (TKE) will therefore not converge until DNS resolution is reached. This makes it difficult to judge whether the increased resolution contributed to an increased quality of the simulation. This leads to the practice that the choice of a grid resolution is mainly based on available computational power. This further leads to studies with a very low mesh resolution of under one million cells (Salim et al. 2011) or up to one billion cells (Phuc et al. 2014).
Non-stationary simulations such as LES, but also DNS, require time dependent conditions at the inlet of the computational domain.

2.6.4 Turbulent Inlet Boundary Condition

If the flow in a urban situation is to be simulated, the flow in the entire domain is usually fully turbulent. This requires the flow to be already turbulent at the inlet. The existing methods can be broadly separated into four categories: laminar-turbulent transition, cyclic domain, driver domain and synthetic turbulence.

The transition method starts with a laminar flow that transitions into turbulent flow. The domain has to be long enough for the flow to develop the desired boundary layer thickness. For thick boundary layer this method is impractical. This is due to the high resolution needed for an accurate boundary layer simulation and large cell count resulting from the long domain length. Even if the turbulence is geometrically tripped and the development section is equipped with roughness elements, a long inlet fetch is needed. This situation is comparable to the wind tunnel setup described in Figure 2.8. This approach was recently used by Phuc et al. (2014) and it is believed that the high cell count of $1,000,000,000$ is mostly due to the simulation of the entire wind tunnel like development section.

The cyclic domain method uses streamwise periodic boundary conditions. The flow is driven by either a streamwise pressure gradient or a fixed velocity at the top of the domain. A cyclic domain represents a streamwise homogeneous situation, as it infinitely repeats the content of the domain. It is therefore widely used for ABL simulations over homogeneous terrain (e.g. Moeng 1984; Sullivan et al. 1994; Beare et al. 2006) or for idealized urban morphologies such as regularly spaced street canyons (e.g. Liu and Barth 2002; Cui et al. 2004; Letzel et al. 2008; Cheng and Liu 2011) or arrays of obstacles (e.g. Cheng et al. 2003; Xie and Castro 2006; Moonen et al. 2012a). Figure 2.12 shows a cyclic domain denoted as driver domain, with a regular array of roughness arrays.

This leads to a third method that is an extension of the cyclic method. Instead of using only a cyclic domain, the flow is extracted from this domain and mapped to the inlet of a subsequent domain. This situation is presented in Figure 2.12, where the cyclic domain is homogeneous...
Figure 2.12: Inflow generation for LES using a cyclic driver domain.

and used as a driver domain and the main domain can be therefore be spatially inhomogeneous. This method was proposed by Lund et al. (1998). He includes a rescaling step in the driver domain, to better control the generated boundary layer. The concept of rescaling was originally devised by Spalart and Leonard (1987) to be able to simulate a spatially growing boundary layer in a cyclic domain. Different mapping methods are described in Baba-Ahmadi and Tabor (2009). The cyclic driver domain method (usually without the added complexity of the rescaling operation) gives realistic turbulence and is applicable to study urban situations (Nozu et al. 2015). The cyclic domain has to be long enough to not influence the statistics of the results negatively through the periodicity generated by the recycled turbulent structures.

A variant of the driver domain method is the database method. Instead of running the driver domain and the main domain simultaneously, slices of inlet data are stored. These are then read from disk and applied to the inlet during the main simulation. This can even be done with measurement data such as stereo-PIV (Maruyama et al. 2012). The finite amount of inlet data however limits the runtime of the main simulation and the large storage capacities required for sufficiently high spatial and temporal resolution limit the applicability of this method.

The last class of methods comprise of the synthetic inflow generators. The development of these methods are mainly motivated by two things. First, by removing the driver domain, a substantial decrease in computational cost can be achieved. Secondly, the turbulence generator allows for precise control of the mean flow profile and the turbulent statistics at
the inlet of the domain. Three principal approaches to generate synthetic
turbulence were identified: spectral, vortex and filtered noise methods.
Spectral methods are based on the inverse Fourier transform of a pre-
scribed spectrum, developed by Lee et al. (1992) and Smirnov et al. (2001).
Using a random phase angle, a turbulent inlet field is synthesized. Vor-
tex methods are based on the superposition of flow fields generated by
discrete vortices, so called turbulent spots. The flow field of a turbulent
spot is expressed by an analytic expression that follows specified correla-
tions. A number of turbulent spots is randomly distributed in a virtual
volume in front of the inlet and moved through this volume by a specified
mean velocity. A two dimensional slice is extracted and imposed at the
inlet. This method and variants thereof were developed e.g. by Sergent
(2002), Jarrin et al. (2006), Mathey et al. (2006), Kornev and Hassel (2007a),
Kornev and Hassel (2007b), and Poletto et al. (2013). The filtered noise
method is based on a random velocity field. Turbulence that is purely
random however quickly dissipates as no coherent structures are present.
The filtered noise method therefore add spatial and temporal correlations
to the random field by the use of a filter function, hence the name "filtered
noise." The method was initially proposed by Klein et al. (2003) and further
developed by Mare et al. (2006), Xie and Castro (2008), and Kim et al.
(2013).

Common to all synthetic methods is the property that the specified inlet
velocity field creates artificial turbulent structures. Inside the simulation
domain, these artificial eddies are modified through the production and
dissipation mechanisms associated with turbulence as described by the
energy cascade (Fig 2.2). This requires the domain to include a develop-
ment section in which this process takes place. After the development
section, a realistic turbulent spectrum is recovered. This process mod-
difies the flow statistics specified at the inlet, for example by a loss of
the specified turbulent energy or a change of the length scales. In addi-
tion, the artificial turbulent field might not be divergence free. Such an
imbalance is corrected by the flow solver through the pressure term in
the Navier-Stokes equations, resulting in pressure fluctuations near the
inlet. Pressure fluctuations are not desirable when building pressures are
studied. Modifications to the inlet velocity field can correct for divergence
and significantly reduce pressure fluctuations, as was demonstrated for a
vortex method (Poletto et al. 2013) and a filtered noise method (Kim et al.
2013; Xuan and Iizuka 2013). Despite the aforementioned shortcomings,
the potential of synthetic turbulence generators for urban type flows was demonstrated through applications by Xie and Castro (2008), Gousseau et al. (2012), Boppana et al. (2012), and Moonen et al. (2013).

Further detailed comparisons between methods are found in Tabor and Baba-Ahmadi (2010), Pronk and Hulshoff (2012), and Dietzel et al. (2014).

2.7 SUMMARY AND NEED FOR FURTHER RESEARCH

Urban flows feature a range of different turbulent flow phenomena, such as turbulent wake flows, shear layers and vortex shedding. These coherent flow structures are generated by the buildings who act like bluff bodies with sharp edges. In built up areas, interaction between buildings lead to complex flow fields, with large vortices in street canyons and shear layers spanning from rooftop to rooftop. Shear layers are unsteady and produce vortices and turbulence.

The current understanding of the exchange of air, heat and pollutants between the street level and the flow aloft recognize the importance of the turbulent shear layer, as it is placed at the interface between the street level and the flow aloft. Wind tunnel testing is widely applied to investigate environmental flows in urban settings. Imaging techniques such as PIV have been used to investigate flow statistics and the spatial structure of turbulence found in street canyons and cavities. Time resolved computational tools such as LES are used to study the interaction of coherent structures in the shear layer and the transport of pollutants. Both methods, PIV and LES, are capable of capturing not only the time averaged statistical properties, but also the time dependent spatial behaviour of turbulent flows.

A limitation in wind tunnel experiments is found to be the lack of time resolved stereoscopic flow field measurements. Numerous studies investigate the turbulence statistics and mean flow properties of street canyons. However, very few experimental studies focus on the spatio-temporal behaviour of the shear layer. Exceptions are Haigermoser et al. (2008) and Bian et al. (2010) who used time resolved PIV. Only planar PIV is used, resulting in two velocity components. Due to the missing third component, TKE could not be accurately measured. The experiment of Bian et al. (2010) features only a thin turbulent boundary layer (0.22 δ/H), whereas Haigermoser et al. (2008) features a thick turbulent boundary...
layer \((2.1 \delta/H)\). The Reynolds number with regard to the cavity height is relatively low \((Re_H = 4625)\). Both experiments use a wide cavity with an aspect ratio from 3-4 (width-to-height). This places these experiments in the wake-interference regime (Fig. 2.6). Wind tunnel experiments are an established tool and the PIV method can be considered mature. In contrast, LES for urban applications however is still under development and is becoming more accessible by the increasing computational power. There is currently a lack of guidelines for urban LES. Therefore, the quality of studies conducted with LES is inconsistent as the results depend on the size of the computational grid. The lack of computationally efficient, proven methods to generate turbulent inflow conditions for urban flows add to this problem. Methods based on artificial turbulence generation were developed, but few actual implementations and codes are available. This leads to few studies that focus on the applicability of artificial turbulence methods to urban flows. The filtered noise inflow generator shows potential for urban flows, but a formal definition of the filter coefficients is missing and the influence of the methods parameters is not well understood.

In this thesis, the street canyon flow features (cavity vortex and shear-layer) found in the skimming flow regime will be studied using time resolved stereo-PIV and LES. Both methods are capable of capturing not only the time averaged statistical properties, but also the time dependent behaviour of turbulent flows. A wind tunnel study is conducted that subjects a cavity to a wide range of turbulent inflow conditions. This allows to study the spatio-temporal behaviour of the shear layer. A model for the urban air is developed based on LES. The quality of the model is demonstrated by performing a grid study and compared to PIV measurements. The filtered noise method is further developed by providing a formal definition for the filter coefficients and the streamwise correlation. The implementation is made available for the open source flow solver OpenFOAM. The capability of the filtered noise method is assessed by a detailed comparison to the wind tunnel measurements.
BACKGROUND

This chapter provides the theoretical background for this thesis. The governing equations are described in Section 3.1. Section 3.2 details the statistical description of turbulence. Section 3.3 describes the time series analysis concepts used and Section 3.4 the additional flow diagnostic methods.

3.1 GOVERNING EQUATIONS

For all conservation equations, the differential form in a Cartesian coordinate system is used. $x_i$ are the coordinates where $i = (1, 2, 3)$ for the three directions $(x, y, z)$. The Einstein summation convention applies whenever an index appears twice in a term.

For the velocity components the notation $u_1, u_2, u_3$ and $u_x, u_y, u_z$ are used interchangeably. For the covariances only the integer subscript is used.

The governing equation for an incompressible Newtonian fluid comprise of the continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0,$$

and the momentum equation:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2},$$

where $u_i$ is the velocity, $p$ the pressure, $\rho$ the density and $\nu$ the kinematic viscosity. These equations are also known as the Navier-Stokes equations. The terms on the left hand side represent the change of momentum over time and the change of momentum through convection. The terms of the right side are the pressure gradient and the diffusion term.

The differential form of the Navier-Stokes equations applies to an infinitesimal fluid particle, whereas the integral formulation applies to a control volume. The integral formulation is useful for the Finite Volume
Method (FVM) used in Computational Fluid Dynamics (CFD) and can be derived either directly from a control volume or by using Gauss’ divergence theorem and the differential form. No detailed presentation of the FVM is made and the reader should refer to Ferziger and Peric (2002).

An additional transport equation can be derived for a conserved passive scalar quantity $\phi$

$$\frac{\partial \phi}{\partial t} + u_i \frac{\partial \phi}{\partial x_j} = D \frac{\partial^2 \phi}{\partial x_j^2} + S \quad (3.3)$$

where $D$ is the molecular diffusivity and $S$ a source term.

3.2 STATISTICAL DESCRIPTION OF TURBULENCE

A signal from a turbulent flow can be described as a stochastic process. In the case where the flow is stationary, the stochastic process is Wide-Sense Stationary (WSS), or weakly stationary. A WSS process requires that the mean does not change over time and the autocovariance only depends on the lag but not the time.

Stationary flows are typically found in continuously running wind tunnels and CFD simulations with constant boundary conditions. Urban flows, or any atmospheric flow for that case, are non-stationary since the conditions are constantly changing, such as the day/night cycle, synoptic scale weather, seasonal changes and changes in atmospheric and ground composition. Figure 3.1 shows the distribution of spectral power for these such effects. These changes are happening on a timescale that is significantly longer than the time scale of turbulence. This makes turbulence a stationary phenomena over the duration of minutes to hours, meaning that the statistical description of turbulence can be applied. The gap between diurnal cycle and turbulence can be explained by the fact that phenomena at this timescale are not periodic but intermittent and are therefore not represented by the spectral power.

It is important to note that even if the time scales of interest are short, it is still required to acquire enough independent samples in order to reach sufficient accuracy of the statistical analysis (see Chapter 4.3.3). For wind tunnel experiments, or CFD simulations, this is only a technical issue, whereas the problem is inherent for field measurements.
Figure 3.1: Time scales of atmospheric turbulent flow, adapted from Van der Hoven (1957)

3.2.1 First and Second Order Moments

The first and second moment, expected value ($E$) and variance ($Var$) respectively, are given by

$$E[X] = \frac{1}{N} \sum_{k=0}^{N} X_k$$

and

$$Var[X] = \frac{1}{N} \sum_{k=0}^{N} (X_k - E[X])^2.$$ (3.5)

The standard deviation $\sigma$ is the square root of the variance:

$$\sigma = Std[X] = \sqrt{Var[X]}$$ (3.6)

The covariance is defined by

$$Cov[X,Y] = \frac{1}{N} \sum_{k=0}^{N} (X_k - E[X])(Y_k - E[Y])$$ (3.7)

The instantaneous velocity vector can decomposed into a mean and fluctuating part:

$$\mathbf{u}(x,t) = \bar{\mathbf{u}}(x) + \mathbf{u}'(x,t)$$ (3.8)
This is generally known as the *Reynolds Decomposition*. Substituting the variable $X$ in Eq. 3.4 and 3.5 with the velocity component $u_i$ we can write the mean velocity and velocity variance as:

\[ \bar{u}_i = \frac{1}{N} \sum_{k=0}^{N} u_{i,k} \]  \hspace{1cm} (3.9)

and

\[ \overline{u_i^2} = \frac{1}{N} \sum_{k=0}^{N} (u_{i,k} - \bar{u}_i)^2. \]  \hspace{1cm} (3.10)

The covariance (Eq. 3.7) is then given by

\[ \overline{u_i' u_j'} = \frac{1}{N} \sum_{k=0}^{N} (u_{ik} - \bar{u}_i)(u_{jk} - \bar{u}_j) \]  \hspace{1cm} (3.11)

and the covariance matrix of the velocity vector $\mathbf{u}$ can be written as:

\[ Cov[\mathbf{u}, \mathbf{u}] = \mathbf{R} = \begin{bmatrix}
    \overline{u_1'^2} & \overline{u_1'u_2'} & \overline{u_1'u_3'} \\
    \overline{u_2'u_1'} & \overline{u_2'^2} & \overline{u_2'u_3'} \\
    \overline{u_3'u_1'} & \overline{u_3'u_2'} & \overline{u_3'^2}
\end{bmatrix} \]  \hspace{1cm} (3.12)

This matrix is known as the *Reynolds stress tensor* $\mathbf{R}$. The covariance matrix, and therefore the Reynolds stress tensor, is symmetric and positive semidefinite.

The significance of the Reynolds stress tensor becomes apparent when looking at the Reynolds-Averaged Navier-Stokes (*RANS*) equations.

### 3.2.2 *RANS Equations*

The averaging operation (Eq. 3.9), can be applied to the Navier-Stokes equations (Eq. 3.1 and 3.2). The results are governing equations for the mean flow field. The continuity equation for the mean velocity reads:

\[ \frac{\partial \bar{u}_i}{\partial x_i} = 0, \]  \hspace{1cm} (3.13)

and taking the mean of the momentum equation gives

\[ \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \frac{\partial \bar{u}_i' u_j'}{\partial x_j} \]  \hspace{1cm} (3.14)
These equations are known as the RANS equations. They are largely identical (but with averaged values) to the Navier-Stokes Equations, only differing by the last term in Equation 3.14. This term arises through the averaged non-linear advection term $\frac{\partial \bar{u}_i u_j}{\partial x_j}$ and when using the Reynolds decomposition (Eq. 3.8), contains the Reynolds stresses $\bar{u}'_i u'_j$. This term is responsible for the transport of momentum through turbulent fluctuations and is magnitudes larger then the molecular diffusion term. In a horizontal, fully developed boundary layer, the turbulent diffusion is therefore the only relevant term for vertical and lateral mixing.

**Scalar Transport**

Analogously, a mean transport equation for a passive scalar can be derived from Eq. 3.3:

$$\frac{\partial \bar{\phi}}{\partial t} + \bar{u}_j \frac{\partial \bar{\phi}}{\partial x_j} = D \frac{\partial^2 \bar{\phi}}{\partial x_j^2} - \frac{\partial \bar{u'}_j \phi'}{\partial x_j} + S$$

(3.15)

The term $\bar{u'}_j \phi'$ is called the turbulent scalar flux. As it cannot be expressed in terms of mean flow quantities, a common assumption is to model it proportionally to the mean scalar gradient:

$$\bar{u'}_j \phi' = D_t \frac{\partial \bar{\phi}}{\partial x_j}$$

(3.16)

and is therefore called the gradient diffusion hypothesis, where $D_t$ is the turbulent diffusivity.

**Turbulent Kinetic Energy**

The Turbulent Kinetic Energy (TKE) is defined as the trace of the Reynolds stress tensor:

$$\kappa = \frac{1}{2} (\bar{u'}_i u'_i)$$

(3.17)
A system of transport equations can be defined for the Reynolds stress tensor. Taking the trace of this system of equations results in the transport equation for $\kappa$:

\[
\frac{\partial \kappa}{\partial t} + \bar{u}_j \frac{\partial \kappa}{\partial x_j} = -\bar{u}_i'\bar{u}_j' \frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial}{\partial x_j} \left[ \frac{1}{2} \bar{u}_i'\bar{u}_j' + \frac{1}{\rho} \bar{u}_i'p' - \nu \frac{\partial \kappa}{\partial x_j} \right] - \nu \frac{\partial \bar{u}_i'\bar{u}_i'}{\partial x_j} \frac{\partial \bar{u}_i}{\partial x_j}
\]

The full derivation can be found in Wilcox (2002).

The production of $\kappa$ appears in the $\kappa$ equation (Eq. 3.18) as a source term. This represents the kinetic energy transfer from the mean flow to the turbulent fluctuations.

The turbulence intensity relates the standard deviation of the velocity to the mean velocity as:

\[
I = \frac{\sigma_u}{|\bar{u}|} = \frac{\sqrt{\bar{u}^2}}{|\bar{u}|}
\]

and gives an indication of the degree of turbulence in a flow.

3.2.3 Convolution and Spectra

The convolution of two functions $f$ and $g$ is defined as:

\[
f \ast g = \int_{-\infty}^{\infty} f(\tau)g(t - \tau)d\tau
\]

The convolution theorem states that:

\[
\mathcal{F} \{ f \ast g \} = \mathcal{F} \{ f \} \mathcal{F} \{ g \}
\]

where $\mathcal{F}$ is the Fourier transform. Applying the inverse Fourier transform $\mathcal{F}^{-1}$ to Equation 3.22, it can be written as:

\[
f \ast g = \mathcal{F}^{-1} \{ \mathcal{F} \{ f \} \mathcal{F} \{ g \} \}
\]

The cross-correlation is defined as:

\[
f \ast g = \int_{-\infty}^{\infty} f^*(\tau)g(t + \tau)d\tau
\]
3.2 Statistical Description of Turbulence

where \( f^* \) is the conjugate complex of \( f \). The cross-correlation theorem reads:

\[
f \ast g = \mathcal{F}^{-1} \{ \mathcal{F}^* \{ f \} \mathcal{F} \{ g \} \}
\] (3.25)

If \( f = g \), \( f \ast f \) is called the autocorrelation and is defined as:

\[
f \ast f = \int_{-\infty}^{\infty} f^*(\tau) f(t+\tau) d\tau
\] (3.26)

and the cross-correlation theorem (Eq. 3.25) reduces to:

\[
f \ast f = \mathcal{F}^{-1} \{ \mathcal{F}^* \{ f \} \mathcal{F} \{ f \} \} = \mathcal{F}^{-1} \{ |\mathcal{F} \{ f \}|^2 \}
\] (3.27)

This is also called the Wiener-Khinchin Theorem.

The term \( |\mathcal{F} \{ f \}|^2 \) on the rhs of Eq. 3.27 corresponds to the definition of the Power Spectral Density (PSD):

\[
\Phi(\omega) = |\mathcal{F} \{ f(t) \}|^2
\] (3.28)

Therefore the autocorrelation and the PSD form a Fourier transform pair:

\[
f(t) \ast f(t) \Leftrightarrow \Phi(\omega)
\] (3.29)

3.2.4 Correlation and Integral Scale

The correlation is closely related to the covariance (Eq. 3.11) and illustrates the correlation of a variable with respect to a temporal lag or a spatial separation. The correlation is usually reported in normalized form: the correlation coefficient is the correlation normalized by the variance. Most frequently, the autocorrelation coefficient and the two-point correlation coefficient is used.

The autocorrelation coefficient is defined as

\[
\rho_i(\tau) = \frac{u'_i(t,x)u'_i(t+\tau,x)}{u'^2_i}
\] (3.30)

where \( \tau \) is the temporal lag. The autocorrelation coefficient is symmetric:

\[
\rho(\tau) = \rho(-\tau).
\] (3.31)

and lies between 1 (perfect correlation) and -1 (perfect anti-correlation).
The two-point correlation coefficient is defined similarly to the autocorrelation, but using a spatial separation vector \( r \) instead of a temporal lag:

\[
\rho_i(r) = \frac{u'_i(t, x)u'_i(t, x + r)}{\sqrt{u'_2(x)} \sqrt{u'_2(x + r)}}
\] (3.32)

or with respect to a specific coordinate direction \( e_j \) with separation distance \( r \):

\[
\rho_{j,i}(r) = \frac{u'_i(t, x)u'_i(t, x + r e_j)}{\sqrt{u'_2(x)} \sqrt{u'_2(x + r e_j)}}
\] (3.33)

The lagged two-point correlation coefficient includes both a temporal lag and a spatial separation and is defined by:

\[
\rho_i(\tau, r) = \frac{u'_i(t, x)u'_i(t + \tau, x + r)}{\sqrt{u'_2(x)} \sqrt{u'_2(x + r)}}
\] (3.34)

The integral time scale is defined as the integral of the autocorrelation coefficient \( \rho(\tau) \) (Eq. 3.30):

\[
T_i = \int_{0}^{\infty} \rho_i(\tau) d\tau
\] (3.35)

and is visualized in Fig. 3.2.

![Figure 3.2](image)

**Figure 3.2:** Integral time scale \( T \), defined as the area under the autocorrelation curve.

The integral length scale is defined similarly to the integral time scale (Eq. 3.35). For the length scale, the two-point correlation coefficient (Eq. 3.33) is integrated over the spatial separation \( r \) (Fig. 3.3). The
two-point correlation function is not necessarily symmetric, especially for non-homogeneous flows. The integral length scale can therefore be defined for the positive direction by:

\[ L_{j,i}^+ = \int_0^\infty \rho_{j,i}(r)dr \]  \hspace{1cm} (3.36)

and for the negative direction by:

\[ L_{j,i}^- = \int_{-\infty}^0 \rho_{j,i}(r)dr \]  \hspace{1cm} (3.37)

For flows that are homogeneous, the average of \( L_{j,i}^- \) and \( L_{j,i}^+ \) can be used to get a single length scale:

\[ L_{j,i} = \frac{1}{2}(L_{j,i}^- + L_{j,i}^+) \]  \hspace{1cm} (3.38)

![Figure 3.3: Integral length scale \( L \), defined as the area under the two-point correlation curve.](image)

The integral time scale can be converted to a length scale by multiplying by a velocity:

\[ L_i = u_c T_i \]  \hspace{1cm} (3.39)

where \( u_c \) is the velocity at which turbulent structures are being convected past a fixed observer (or location). The convection velocity can be obtained by using the lagged two-point correlation (Eq. 3.34). For a fixed separation distance the time lag at which the correlation coefficient is maximal is computed and from that (and the known separation distance) the convective velocity is obtained. This requires simultaneously sampled velocity time series data at two locations and a with a high sample frequency (to determine the location of the maximum). A more simple
approach is to use *Taylor’s hypothesis of frozen turbulence* (Taylor 1938). This assumes that turbulent eddies do not change over a small distance, they are essentially *frozen* and are transported past the observer by their mean velocity. The length scale is then obtained by:

\[ L_i = \bar{u}T_i \]  

(3.40)

### 3.3 Time Series Analysis

Time series analysis was established as a field in statistics during the 1940s and 1950s (Calder and Davis 1997) and is concerned with the statistical modelling and analysis of time series data. A statistical method used is the class of Autoregressive Moving-Average (ARMA) models. The ARMA model describes a time series by a linear combination of past values (auto-regressive) and by a weighted average of random numbers (moving average). A detailed description can be found in Box et al. (2008).

The complete ARMA process is given by:

\[
X_t = c + \varepsilon_t + \sum_{i=1}^{p} \phi_i X_{t-i} + \sum_{i=1}^{q} \theta_i \varepsilon_{t-i}
\]

(3.41)

consisting of a constant \( c \), an Autoregressive (AR) part of order \( p \) and a Moving-Average (MA) part of order \( q \). \( \phi_i \) and \( \theta_i \) are the model parameters. The term \( \varepsilon_t \) is a normally distributed random innovation term with zero mean and variance \( \sigma^2 \) (white noise).

After the identification or choice of model order \( p \) and \( q \), the model parameters \( \phi_i \) and \( \theta_i \) are determined to best represent existing data. Different methods for model fitting are described in Box et al. (2008).

Instead of fitting a model to existing time series data, the model parameters can also be determined from desired statistical properties. This is presented for the AR(1) model (AR(p) model of order one). From 3.41, the AR(1) model reads:

\[
X_t = \phi X_{t-1} + \varepsilon_t.
\]

(3.42)

with \( \phi \) being the AR(1) parameter and \( \varepsilon_t \) the innovation term. As derived e.g. in Hamilton (1994), the variance of an AR(1) process is given by
3.3 Time Series Analysis

\[
\sigma_X^2 = \frac{\sigma^2}{1 - \phi^2},
\]

(3.43)

and the Autocorrelation Function (ACF) as:

\[
\rho(\tau) = \phi^\tau, \quad (3.44)
\]

with an exponential shape. The AR(1) parameter \(\phi\) is therefore simply the autocorrelation at lag \(\tau = 1\). Time series of turbulent velocity data show an exponentially decaying autocorrelation (e.g. Pope 2004), making the AR(1) model interesting to generate synthetic time traces of turbulent velocity signals.

By further applying the definition of the integral time scale (Eq. 3.35):

\[
T = \int_0^\infty \phi^\tau d\tau = \left. \frac{\phi^\tau}{ln(\phi)} \right|_0^\infty = -\frac{1}{ln(\phi)}
\]

(3.45)

the AR(1) parameter \(\phi\) can be expressed in terms of the integral time scale \(T\):

\[
\phi = e^{-\frac{1}{T}}.
\]

(3.46)

The AR(1) model can therefore generate velocity signals with a given variance and integral time scale. This is illustrated in Figure 3.4 by two time series generated by an AR(1) model. The mean and variance was kept the same (\(X_t = 0\) and \(\sigma_X^2 = 1.0\)) and an identical series of random innovation terms \(\varepsilon_t\) was used. The first realization was generated with an integral timescale of \(T = 2.0\) (Fig. 3.4a) and the second with an integral timescale of \(T = 20.0\) (Fig. 3.4b). Both time series show small scale fluctuations over short time spans. This is due to the exponential decay of the ACF. The longer integral time scale of the second time series is visible by the additional fluctuations varying over a longer time span.
Figure 3.4: Two realizations of an AR(1) model with a zero mean and a variance of one, for an integral time scale of (a) $T = 2.0$ and (b) $T = 20.0$.

3.4 Flow Diagnostic

3.4.1 Quadrant Analysis

Quadrant analysis is based on the classification of the instantaneous velocity fluctuations. Wallace et al. (1972) associates the sign of the fluctuating velocity components to events as listed in Table 3.1. An ejection is a low momentum fluid parcel transported upwards into higher momentum flow, whereas a sweep event corresponds to a high momentum parcel transported downwards into lower momentum flow. An interaction can be understood as a fluid parcel returning to its previous location, meaning a low momentum parcel returning to a low momentum flow (inward interaction) and a high momentum parcel returning to high momentum flow (outward interaction). In a boundary layer, ejection and sweep events occur predominantly, as only they contribute to the negative shear stress $u_1' u_2'$. Interactions are less frequent.

3.4.2 Vortex Identification

A coherent structure in turbulent flow can be identified by its vortex core. To detect the vortex core, one method used is to study the invariants of
Table 3.1: Definition of the four quadrants in a boundary layer. $u_1$ is the horizontal velocity, $u_2$ the vertical velocity (wall normal).

<table>
<thead>
<tr>
<th>Event</th>
<th>$u'_1 &lt; 0$</th>
<th>$u'_2 &gt; 0$</th>
<th>$u'_1 &gt; 0$</th>
<th>$u'_2 &lt; 0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ejection</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sweep</td>
<td>$u'_1 &gt; 0$</td>
<td>$u'_2 &lt; 0$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Outward</td>
<td>$u'_1 &gt; 0$</td>
<td>$u'_2 &gt; 0$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interaction</td>
<td>$u'_1 &lt; 0$</td>
<td>$u'_2 &lt; 0$</td>
<td>$u'_1 &lt; 0$</td>
<td>$u'_2 &lt; 0$</td>
</tr>
</tbody>
</table>

the velocity gradient tensor, allowing to distinguish between sheering flow and rotating flow. The vortex identification criterion $Q$ (Hunt et al. 1988) is defined by:

$$Q = \frac{1}{2} (||\Omega||^2 - ||S||^2) > 0$$

(3.47)

where $S = \frac{1}{2} (\partial u_i / \partial x_j + \partial u_j / \partial x_i)$ is the rate-of-strain tensor and $\Omega = \frac{1}{2} (\partial u_i / \partial x_j - \partial u_j / \partial x_i)$ the vorticity tensor. A vortex can be identified by positive $Q$. For Particle Image Velocimetry (PIV) data acquired on a streamwise plane, the full velocity gradient tensor is not available. The two dimensional simplification is defined as:

$$Q^{2D} = \frac{1}{2} \left( 2 \frac{\partial u_x}{\partial y} \frac{\partial u_y}{\partial x} - \frac{\partial u_x^2}{\partial x} - \frac{\partial u_y^2}{\partial y} \right) > 0.$$  

(3.48)

$Q^{2D}$ will only capture a projection of the vortex axis on the out-of-plane axis. $Q^{2D}$ is therefore identical to $Q$ for a vortex normal to the PIV plane and zero when the axis is parallel.

3.5 CONCLUSION

This chapter presented the governing equation for the flows investigated in this thesis. Furthermore, the statistical concepts and tools to describe turbulence were presented. A short introduction to time series analysis was given. The statistical tools and the presented diagnostic tools (vortex core identification and quadrant analysis) will be used to investigate the wind tunnel measurements presented in the next chapter.
CAVITY FLOW EXPERIMENT

This chapter contains the main experimental part of the thesis: the wind tunnel experiment conducted on a unit aspect ratio cavity under perpendicular flow, using time resolved stereo Particle Image Velocimetry (PIV).

A measurement setup was developed with the purpose of:

- Studying the effect of inflow turbulence on the cavity flow, specifically the shear layer.
- Providing high resolution, three-component time resolved flow data for the verification of Large Eddy Simulations (LES)

These two aims influenced the design choices of the measurement setup. The setup was developed simultaneously with the flow simulation, specifically with comparability in mind. This motivated the choice of the PIV measurement plane and the field of view.

Additionally, the choice of a split floor allows to vary the degree of turbulence upstream of the cavity to the extent of having laminar flow. A boundary layer profile according to norms (e.g. VDI 3783 2000; ESDU 85020 2001) is a generalized representation of an Atmospheric Boundary Layer (ABL) and therefore a representation of generalized urban geometry. In reality, a street canyon might be located in the wake flow of a larger upstream obstacle and the flow encountered by the street canyon might therefore be quite specific. For this reason, a large number of turbulence generating obstacles (barriers and spires) were tested to create specific boundary layers. This allows to find interesting and distinct flow cases.

This chapter is divided into seven main sections: The wind tunnel facility, measurement equipment and model geometry is described in Section 4.1. The measured cases are described in Section 4.2. An estimation of the uncertainty for the time averaged and the time resolved PIV measurements is provided in Section 4.3. Results of hot wire measurements of the wake flows of turbulence generating spires are presented in Section 4.4. The influence of turbulence on the cavity vortex is described in Section 4.5. Three selected cases, representing characteristic shear layer conditions, are
CAVITY FLOW EXPERIMENT

analyzed in detail in Section 4.6, followed by a discussion and conclusion in Section 4.7

4.1 EXPERIMENTAL SETUP

4.1.1 Experimental Facility

Measurements were conducted at the ETHZ/Empa ABL wind tunnel, a closed circuit Göttingen-type boundary layer wind tunnel (Fig. 4.1). The test section is closed and 10.4 m long, 1.9 m wide and 1.3 m high. Large windows and a glass ceiling maximize optical access. The fan has a nominal power of 110 kW and is capable of generating free stream velocities from 0.5 up to 25 m/s in the test section. Flow seeding is provided by a system that generates 1 µm Di-Ethyl-Hexyl-Sebacat (DEHS) aerosol particles. The tunnel can also be operated in open-loop mode. The open loop mode is useful for rinsing the ABL section from the seeding particles.

Figure 4.1: ETHZ/Empa atmospheric boundary layer wind tunnel.
Two traverse systems are installed in the last 2.6 m of the test section. A large traverse around the test section can move streamwise along the test section and hold heavy equipment, such as cameras and lasers. Inside the test section, a light-weight, stepper motor driven traversing system is installed, allowing a probe to be moved within a 2.3 m x 0.3 m x 1.6 m volume with a step size of 1/100th mm. The linear positioning elements Rose+Krieger DuoLine Z 50 are used for all axes and feature a timing belt driven guide table with a repetition accuracy of 0.05 mm and a positioning accuracy of 0.1 mm/300 mm. Induction switches allow to perform a reference drive to reliably set the origin of the traverse system. The traverse system is also used to define an absolute coordinate system in the test section, allowing for repeatable wind tunnel model placement and accurate PIV laser sheet alignment.

4.1.2 Measurement System

The principal measurement system used is a time-resolved stereoscopic PIV system from LaVision and is composed of two 12 bit CMOS cameras and a dual cavity Nd:YLF laser with a wavelength of 527 nm and a maximum energy per pulse of 30 mJ (at 1 kHz repetition rate and a pulse duration of 150 ns). The cameras have a maximum resolution of 2016 x 2016 pixels at a recording frequency of up to 640 Hz. The 36 GB of memory allow storage of 3155 double-frame images at full resolution. The two cameras were mounted at opposite sides of the wind tunnel test section, perpendicular to the flow and tilted down at an angle of 33° from the horizontal (Figure 4.2). The two cameras have to be set up at an angle to each other, in order for the stereoscopic reconstruction to work. The vertical V-shaped setup, located in the plane of the cavity, was chosen to have uninterrupted optical access to the cavity model. The optics consisted of 135 mm F2.0 Canon objectives on a Scheimpflug mount. The light sheet, generated by a cylindrical lens above the wind tunnel, was vertically aligned with the test section center-plane.

Additionally, a Dantec hot wire system in combination with the traverse was used to sample large field turbulent flow statistics. The probe used was a Dantec 55P16 Constant Temperature Anemometry (CTA) miniature single wire probe with a 1.25 mm long and 5 µm diameter platinum-plated tungsten wire. Velocity signals were acquired using the a MiniCTA
single channel hot wire bridge with analog voltage output and a National Instruments NI USB-9162 A/D converter with a 10 kHz sample rate. Calibration of the probe was performed using a Dantec Stream-Line Pro Automatic Calibrator to get a 5th order calibration polynomial. A correction of the measured voltage signal was performed to account for temperature difference between the calibration temperature and the measurement temperature, prior to the conversion to velocity.

4.1.3 Model Geometry

A split floor was installed in the downstream part of the test section, raised 31.5 cm above the wind tunnel floor (Fig. 4.3). This is well above the wind tunnel boundary layer, which measures around 15 cm in height. The experimental setup is composed of four main sections: adjustable inlet flap, inlet fetch, cavity and outlet plane. The inlet flap can be deflected downwards to prevent flow separation due to the increased blockage of the cavity model. The inlet flaps are manufactured from wire-erosion cut aluminum and have a slightly rough finish. The inlet plate, cavity and outlet plate are manufactured from phenolic coated birch plywood and have a smooth finish. The cavity height is $H_c = 100\ mm$ and the cavity width (width of gap between upstream and downstream edge) measures $W_c = 101.4\ mm$. This results in an aspect ratio of $H_c/W_c = 0.99 \approx 1$. The complete model is 1.8 m wide (spanwise), leaving a 5 cm gap on each side between the model and the wind tunnel wall (Fig 4.2). To minimize the influence of the wind tunnel walls, the cavity ends are closed, using

![Figure 4.2: Wind tunnel cross section showing stereo PIV camera arrangement and imaging plane, with installed cavity model.](image)
thin 500 mm long wooden end caps. They extend 440 mm from the wind tunnel floor to well above the cavity. The model is supported by 3 mm thin sheet metal struts. To reduce aerodynamic drag from the protruding cavity on the lower side of the model, and therefore to increase flow-through, a fairing made from Polystyrene foam and a flexible wooden plate was installed.

A 200 mm wide center section of the cavity has the horizontal surfaces (inlet, floor and outlet) equipped with flush mounted, aluminum coated front surface mirrors. The mirrors have a reflectivity above 94%, reducing unwanted diffusion of laser light. Diffusively reflected laser light from the line where the light sheet hits a model can be brighter than the particles. This is especially problematic for stereo PIV, where due to the angled camera setup the light sheet reflection line is always visible to the camera. A front surface mirror reduces the width of the reflection line and therefore enables near-surface measurements.

A set of spires and barriers was manufactured. Different combinations of spires and barriers were investigated (see further). The spires were

**Figure 4.3:** Wind tunnel setup for 2D cavity model with split floor showing the approach flow, laser sheet and PIV field of view (dashed rectangle). The right cavity end cap is rendered transparent for clarity.
mounted downstream of the inlet flap, at a position $x = 608 \text{ mm}$ upstream of the cavity. They have a triangular shape and resemble Irwin-type spires (Irwin 1981). The spire geometry is shown in Figure 4.4. The spacing between the spires was half the height $H$. The spires are made from 2 mm thin aluminum. A rectangular barrier was placed 55 mm upstream of the spires. The barriers were made from 2 mm thin aluminum and were 1800 mm wide.

4.1.4 PIV Vector Processing and Velocity Post-Processing

Image acquisition and PIV vector processing was performed with the software DaVis 8 from LaVision GmbH. Stereo PIV vector processing was performed on a dedicated cluster with 10 PCs and an attached storage capacity of 80 TB. A multi-pass approach was used with a final interrogation window of 24 pixels, with a 50% window overlap. This resulted in a window size of $dx = dy = 1.574 \text{ mm}$, providing $60 \times 60$ three-component velocity vectors inside the cavity. Post-processing and data evaluation was conducted using custom written Python 2.7 code, utilizing the Numpy$^1$, Scipy$^2$ and the in-house developed pyFlowStat (Immer and Vonlanthen 2015b) modules.

---

1 http://www.numpy.org/
2 http://www.scipy.org/
The cavity flow fields were measured for different configurations of turbulence generating spires and barriers. The spires are defined in Table 4.1a and the barriers in Table 4.1b. The case name is then a combination of the two, e.g. B1.2 15m for a 1.2 cm high barrier infront of a 15 cm, medium wide spire. The case without any spire or barrier is denoted ThinBL, standing for Thin Boundary Layer. The spire geometrical properties are summarized in Table 4.2. The blockage is the amount of area covered by the spire with respect to an area with the height of the spire and the width of the model (1.8 m). Since the spacing of the individual spires is defined as half the height (Fig. 4.4), a constant width-to-height ratio results in a constant blockage, independent of the spire height: 7.7% for the narrow spires, 15.4% for the medium spires and 30.8% for the wide spires.
Table 4.2: Geometrical properties of the turbulence generating spires

<table>
<thead>
<tr>
<th>Case</th>
<th>Spire Height</th>
<th>Spire Width</th>
<th>Blockage</th>
</tr>
</thead>
<tbody>
<tr>
<td>ThinBL</td>
<td>-</td>
<td>-</td>
<td>0%</td>
</tr>
<tr>
<td>5n</td>
<td>50 mm</td>
<td>3.85 mm</td>
<td>7.7%</td>
</tr>
<tr>
<td>5m</td>
<td>50 mm</td>
<td>7.7 mm</td>
<td>15.4%</td>
</tr>
<tr>
<td>5w</td>
<td>50 mm</td>
<td>15.5 mm</td>
<td>30.8%</td>
</tr>
<tr>
<td>10n</td>
<td>100 mm</td>
<td>7.7 mm</td>
<td>7.7%</td>
</tr>
<tr>
<td>10m</td>
<td>100 mm</td>
<td>15.4 mm</td>
<td>15.4%</td>
</tr>
<tr>
<td>10w</td>
<td>100 mm</td>
<td>30.8 mm</td>
<td>30.8%</td>
</tr>
<tr>
<td>15n</td>
<td>150 mm</td>
<td>11.55 mm</td>
<td>7.7%</td>
</tr>
<tr>
<td>15m</td>
<td>150 mm</td>
<td>23.1 mm</td>
<td>15.4%</td>
</tr>
<tr>
<td>15w</td>
<td>150 mm</td>
<td>46.2 mm</td>
<td>30.8%</td>
</tr>
</tbody>
</table>

4.2.1 PIV Flow Measurements

For each case, two image sequences were recorded with the PIV system at a wind tunnel fan speed of 100 rpm. Each sequence consists of 3155 double-frame stereo images with 2016x2016 pixel resolution. The first sequence was recorded at 10 Hz, to provide data of the average flow field, the second sequence at 600 Hz. A separation time of 300 µs between the two frames of each double frame image pair was found to give adequate particle shift values.

The average free stream velocity above the cavity was calculated from time-averaged measurements and was found to be $u_\infty = 3.14 \text{ m/s}$ with a relative standard deviation of 0.96%. With an air temperature of 22.3°C ± 1.5°C and standard atmospheric conditions at 440 m altitude, the Reynolds number with respect to the cavity height $H_c$, was calculated to be $Re_H = 19'260 \pm 175$.

4.3 Uncertainty Estimation

Three main areas of uncertainty have been identified that lead to errors or uncertainty: design of the experiment, PIV measurement uncertainty and statistical uncertainty.
4.3 Uncertainty Estimation

4.3.1 Experiment Design

Unwanted flow effects due to the experimental setup were reduced by careful design. Gaps and small steps between the model parts were sealed using tape and special care was taken upstream of the measurement area.

The effect of open or closed cavity ends on the flow were investigated experimentally. Removed end caps showed a strong superimposed spanwise vortex in the cavity (see higher velocities and Turbulent Kinetic Energy (TKE) in Fig. 4.5), most probably driven by low pressure zones at each cavity end. It is presumed that these are generated by local acceleration of air around the cavity section due to its larger blockage with respect to the split floor. Even a smaller size than the finally installed end caps, one that covers just the cavity ends (120 mm × 140 mm), was found to reduce most of the unwanted vortex.

**Figure 4.5**: Profiles of mean velocity and TKE for the case ThinBL through the cavity center for a) a vertical line at $x = 0.05$ m and b) a horizontal line at $y = -0.05$ m. The solid lines represent two measurements 10 days apart (run 1 and run 2). The dashed line shows the effect of open cavity ends.
Cavity Flow Experiment

The temperature in the wind tunnel room is controlled and only small fluctuations were observed (±1.5°C) between measurement days, minimizing the change of Reynolds number. Repeatability of the flow was demonstrated for two measurements at the beginning and end of the measurement campaign (10 days apart) and is shown in Figure 4.5. Only for TKE slight differences can be observed within the cavity. This difference is believed to be due to statistical uncertainty.

4.3.2 Measurement Uncertainty

PIV uncertainty was reduced by performing the measurements according to best practice (LaVision 2011b). The laser light sheet was geometrically aligned with the wind tunnel centerline, and the camera angles set to 33°. The initial camera calibration was performed prior to installing the model in the tunnel with a precision milled 200 × 200 mm two-level calibration plate. The camera and lens parameters were calculated by fitting a pin-hole model to the calibration images. DaVis reports the standard deviation of the fit to be 0.15 pixel. The registration error was reduced by using a Stereo-PIV self-calibration procedure (Wieneke 2005). No peak locking was evident for either the cavity or the boundary layer flow, suggesting adequate particle image size (LaVision 2011a).

To assess the measurement uncertainty, the sensitivity coefficients $c_i$ for PIV equation $u = \alpha \left( \frac{\Delta X}{\Delta t} \right)$ were calculated. The procedure described in ITTC (2008) was adopted. Table 4.3 presents the results, where $\alpha$ relates to the camera calibration, $\Delta X$ to the pixel displacement, $\Delta t$ to the timing accuracy of the laser. $\Delta X$ was determined using the correlation statistics method (Wieneke 2014), as implemented in the DaVis Software. The estimated overall PIV uncertainty is well below 1%.

4.3.3 Statistical Uncertainty

Statistical uncertainty was assessed according to the procedure described in Nobach and Tropea (2007) assuming a normal distribution. The standard deviation of the estimated mean for the three velocity components can be calculated by using the Standard Error (SE) of the mean

$$SE_{\pi_i} = \sqrt{\frac{R_{ii}}{N_{eff}}}$$

(4.1)


Table 4.3: PIV velocity uncertainty estimation

<table>
<thead>
<tr>
<th>Par.</th>
<th>Error Sources</th>
<th>$u_c$ [m/s]</th>
<th>$c_i$ [px/s]</th>
<th>$c_i u_c$ [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>calibration</td>
<td>6.456E-07</td>
<td>24515.4</td>
<td>0.01582</td>
</tr>
<tr>
<td>$\Delta X$</td>
<td>pixel displacement</td>
<td>0.03</td>
<td>0.4269</td>
<td>0.01281</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>image interval</td>
<td>3.002E-07</td>
<td>-10466.7</td>
<td>-0.00314</td>
</tr>
<tr>
<td>Combined</td>
<td></td>
<td></td>
<td></td>
<td>0.0206 m/s (0.66%)</td>
</tr>
</tbody>
</table>

where $R_{ii}$ is a diagonal component of the Reynolds stress tensor, and $N_{eff}$ the effective sample size. The effective sample size is estimated using

$$N_{eff} = \min(N_{samples}, \frac{T_{meas}}{2 T_i})$$  \hspace{0.5cm} (4.2)

where $T_{meas}$ is the total duration of the measurement, $T_i$ the largest integral time scale and $N_{samples}$ the number of recorded samples. By setting $N_{samples} = N_{eff}$, the required maximum acquisition frequency $f_{acq}$ can therefore be calculated using

$$f_{acq} \leq \frac{1}{2 T_i}$$  \hspace{0.5cm} (4.3)

The case 15w features a high turbulence boundary layer with large coherent structures. With a timescale of $T_x = 0.04$ s in the approach flow boundary layer, the maximum acquisition frequency results in 12.5 Hz. For the time averaged measurements, the chosen total measurement duration is $T_{meas} = 315.5$ s ($f_{acq} = 10$ Hz). The number of effective samples results in the total samples $N_{eff} = \min(3155, 3944) = 3155$. The $SE$ for the largest Reynolds stress of the case 15w ($Re_{11} = 0.2$ $m^2/s^2$) results in $SE_{\bar{u}_x} = 0.8\%$.

To estimate the error of the Reynolds stress, the standard deviation of the estimator is calculated using

$$\sigma_{R_{ii}} = \sqrt{\text{Var}(R_{ii})} = R_{ii} \sqrt{2/(N_{eff} - 1)}$$  \hspace{0.5cm} (4.4)

The relative error of the Reynolds stress therefore only depends on the sample size. With 3155 effective samples, this corresponds to 2.5%.
Uncertainty estimation of the integral time scale

Figure 4.6: Autocorrelation coefficients for $T=0.013s$, comparing ACF realizations of the AR(1) model (gray), the theoretical exponential ACF (orange) and an exemplary PIV measurement (red)

The integral time scale $T$ provides a measure of the persistence and indirectly the size of coherent structures. It is calculated by integrating the autocorrelation function (Eq. 3.30) from zero to infinity (Eq. 3.35). The current memory limitations of high resolution time resolved PIV systems can result in relatively small sample sizes (compared with e.g. hot wire measurements), and therefore short time series. This necessitates a review of the procedure of how to compute the integral time scale from the sample autocorrelation coefficients and to provide an estimate of the uncertainty.

A numerical approach based on time-series analysis was devised to statistically estimate the uncertainty of measured integral time scales. A large sample size of statistically independent, synthetically generated velocity time series data represents repeated PIV measurements. A measurement is approximated using an Autoregressive (AR) model of order one (Eq. 3.42). The Autocorrelation Function (ACF) of the AR(1) model shows an exponential decay (Eq. 3.44), and the AR(1) parameter can be adjusted through the integral time scale (Eq. 3.46) to best match the experiment. The approximate exponential nature of the measured ACF
is shown in Figure 4.6. The number of samples and frequency of the generated data was chosen to match the PIV measurements (3155 samples and $f_{acq} = 600$ Hz). 2000 realizations were generated, each standing for a single PIV measurement. Figure 4.6 shows the results for the 2000 realizations with $T_i = 0.013$ s. The deviations of the calculated autocorrelation coefficients from the exact, exponential ACF is visible as a gray band. The exact correlation approaches zero at about 30-40 lags $\tau$, whereas each predicted ACF from the realizations shows large deviations from zero at higher lags. The 95% confidence interval CI of this noise is well predicted by

$$CI_{95} = \frac{1.96}{\sqrt{N_{eff}}}$$ (4.5)

where $N_{eff} = 242$ is the effective sample size (calculated using Eq. 4.2).

For each realization, the integral time scale was computed using one of four methods (visualized in Fig. 4.7): least squares fitting of an exponential function, extracting the lag $\tau$ at which the correlation drops below $1/e$ ($\rho(\tau) \approx 0.368$), numerical integration up to the first minimum and numerical integration up the zero crossing. Finally, statistical analysis on the computed integral time scales can be performed. The results are presented in Table 4.4.

**Figure 4.7**: Methods to extract the integral time scale $T$ from the autocorrelation coefficient $\rho$. $T$ can be obtained directly at the lag where $\rho$ drops below $1/e$, or by integrating either a fitted curve or by integrating $\rho$ up to a maximum lag $\tau$ (first minimum or zero crossing).

The $1/e$ method performs overall the best, followed by the curve fitting method. The numerical integration shows either a larger bias error (first minimum), or a large standard deviation (zero crossing). Since the result
Table 4.4: Results of AR(1) integral length scale analysis (T = 0.013s)

<table>
<thead>
<tr>
<th>Method</th>
<th>Integral scale</th>
<th>RMS error (%)</th>
<th>Bias (%)</th>
<th>Std (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theoretical</td>
<td>0.013</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Curve fit (exponential)</td>
<td>0.01267</td>
<td>12.8</td>
<td>-2.5</td>
<td>12.5</td>
</tr>
<tr>
<td>1/e</td>
<td>0.01281</td>
<td>10.2</td>
<td>1.5</td>
<td>10.1</td>
</tr>
<tr>
<td>Integration (first minimum)</td>
<td>0.01194</td>
<td>14.5</td>
<td>-8.1</td>
<td>12</td>
</tr>
<tr>
<td>Integration (zero crossing)</td>
<td>0.01322</td>
<td>20.6</td>
<td>1.7</td>
<td>20.5</td>
</tr>
</tbody>
</table>

of the integration is sensitive to the location of the upper limit, these methods only work well with very low uncertainty at the lags where the limit is located. As seen in Fig. 4.6 this is clearly not the case. The locations of the zero crossings and the minima of the simulated ACFs are spread out between $\tau \approx 15$ and $\tau \approx 50$. The $1/e$ method and the curve fitting method are similar, as they both assume an exponential shape of the ACF. The $1/e$ method takes only one point into account ($\rho = 1/e$), located at relatively low lags where the uncertainty of the autocorrelation coefficient is still small. This explains the good performance. The curve fitting method uses all autocorrelation values but can handle the high-lag spurious correlations (noise), as these get averaged out (main disadvantage of the integration methods). Additionally, curve fitting method provides feedback with regard to the goodness of the fit (allows detection of bad PIV data or a significant deviation from the assumed shape). As the methods perform similarly, the optimal choice depends on the required control over the correlation function, in which case the fitting is more adequate, or the robustness and speed, where the $1/e$ method is more suitable.

For the curve fitting method, the uncertainty was computed for a range of integral time scales. The results in Figure 4.8 show that the uncertainty grows with increasing time scale. From this, a simple lookup table can be created for each type of measurement (sample size and acquisition frequency) to apply the uncertainties to the measurements.

With an increasing time scale, the effective sample size $N_{eff}$ reduces (for a constant sample size $N_{samples}$). Eq. 4.5 shows that the only way to reduce the uncertainty is to increase the number of (effective) samples that contribute to the statistical accuracy of the estimated sample
autocorrelation coefficients. However, the two parameter determining the measurement duration, and therefore the effective sample size, \( N_{\text{samples}} \) and \( f_{\text{acq}} \) (Eq. 4.2), are predetermined. The sample size for time resolved PIV (number of acquired images) is set by hardware limitations and the acquisition frequency \( f_{\text{acq}} \) by the requested temporal resolution \( (f_{\text{acq}}/2) \).

To conclude: A method as been presented to estimate the statistical uncertainty of integral time scales for time resolved PIV measurements. This approach takes the numerical method used to integrate the ACF into account, thus includes the error propagation of the autocorrelation coefficient into the integral scale. The method can be applied \textit{a priori}, e.g. to design a wind tunnel measurement or \textit{a posteriori}, to assess the quality of measured data. Results were presented for an exponentially decaying ACF only (AR(1) model), however the procedure can be used for an arbitrary ACF, using a higher order AR(p) model.

4.4 SPIRE FLOW FIELD

To complement the PIV measurements, the flow fields produced by the nine types of spires (Tab. 4.1a) were measured using a hot wire probe.
4.4.1 Hot Wire Measurements

The traverse and the CTA probe was used to acquire samples on a plane. Two planes were measured: a streamwise vertical plane at \( z = 0 \) \( m \) (between two spires) and a horizontal plane at half spire height (\( y = 0.5H_s \)). A rectangular measurement grid was used and adapted for each spire to fit the geometry. A finer resolution was set for the wake region of the spire and a coarser resolution was used further downstream. After each movement to a new grid location, a hold time of 1 second between points on a line and 10 seconds between lines was used, to damp the vibrations of the traverse arm. At each grid location, a maximum of 200’000 points were sampled at a acquisition rate of 10 kHz, resulting in a maximum measurement time of 20 seconds. A run-time computation of the integral time scale was used, in order to stop the measurement if the defined quality criterion for the standard error of the mean (Eq. 4.1) and the relative error of the velocity variance (Eq. 4.4) were reached (\( SE_\pi < 0.2\% \) and \( \sigma_{R11} < 5\% \)). This prevented unnecessary sampling time at points with low or small scale turbulence. On average, each plane took three hours to measure.

4.4.2 Results

The streamwise vertical plane shows the location of the cavity with respect to the wake flow of the spire (Fig. 4.9). As the vertical plane is located between two spires, an initial velocity higher than the free-stream (\( u_\infty = 3.14 \) \( m/s \)) is measured. After some distance downstream, the individual spire wakes interfere, indicated by an increase in turbulence intensity \( I \) (Fig. 4.10). This is especially pronounced for the medium and wide spires. The narrow spires show a more gradual increase of \( I \). The wake flow of the spires are visible in the measured velocity on a horizontal plane (Fig. 4.11), showing a low speed area just behind the spires, indicating separated flow and a high speed jet in between the spires. The wakes of the narrow spires are thin and spaced further apart, delaying the mixing. For the medium spires, the wakes are closer together and fully merge at around \( 2 \times H_s \) downstream of the spire (at \( x = -0.3 \) \( m \) for the spire 15m, at \( x = -0.4 \) \( m \) for the spire 10m and at \( x = -0.5 \) \( m \) for the spire 5m). After the merging location, elongated streaks of velocity are
still visible, however opposite in velocity difference than behind the spire. For the wide spires, the individual spire wakes mix almost immediately behind the spire. The wakes of multiple spires seem to join up and form larger wake fields, extending far downstream.

The turbulence generated in the wake region of the spires is examined further. In Figure 4.12, vertical streamwise planes of turbulence intensity are displayed in a coordinate system normalized by the spire height $H_s$ with origin at the spire base. It is apparent, that the height of the wake region scales with the spire height $H_s$. Furthermore, the width-to-height ratio of the spire scales the shape and length of the wake. Separately for the three width-to-height ratios, the magnitude of the turbulence intensity in the wake is shown in Figure 4.13. It can be seen that the higher the ratio (and therefore the higher the blockage), the better the profiles collapse at each normalized downstream distance. The other spires (narrow and medium) still show some independence at around a third of the spire height. Clear differences in turbulence intensity are apparent near the ground and the tip of the spire.

From these observations, some assumptions can be made about the way the spires generate turbulence. The flow hits the spire frontally and the flow velocity is reduced to zero at the stagnation point. The flow accelerates around the spire and separates from the edges, creating a wake flow behind the spire. The low velocity due to the separation and the high velocity of the free stream generate strong shear layers that produce vortices. Additional vortices are produced due to the turbulent wake flow. For the narrow spires, the wake flow is small and the turbulence generation in the shear layer dominates. For the wide spires, a large wake forms behind the spires. This wake produces large scale turbulent structures that dominate over the shear layer generated turbulence. The flow around the wise spires behaves more like the flow around a bluff body.
Figure 4.9: Streamwise vertical planes of the mean velocity field $\bar{u}$, generated by the nine spires. $x = 0$ is located at the upstream cavity edge.
Figure 4.10: Streamwise vertical planes of the turbulence intensity field $I$, generated by the nine spires. $x = 0$ is located at the upstream cavity edge.
Figure 4.11: Horizontal planes \( (y = H_s/2) \) of the mean velocity field \( \bar{u} \), generated by the nine spires. The spanwise distance \( z \) is normalized by the spire height \( H_s \). \( x = 0 \) is located at the upstream cavity edge.
Figure 4.12: Streamwise vertical plane of turbulence intensity for the nine spires, at $z = 0\, m$ (centerline). The vertical and horizontal distances are normalized by the spire height $H_s$, with the spire located at $x/H_s = 0$.

Figure 4.13: Vertical profiles of turbulence intensity for the nine spires, plotted for different downstream distances (normalized by the spire height $H_s$, with the spire located at $x/H_s = 0$).
4.5 CAVITY VORTEX

The mean cavity flow is dominated by a large clockwise rotating vortex. Figure 4.14 shows the flow topology, with the main vortex, two small vortices in the lower corners and a small vortex by the upstream corner. The upstream boundary layer and profiles of horizontal and vertical velocity through the vortex center are depicted by shaded areas.

**Figure 4.14**: Sketch of the mean cavity vortex, with inflow boundary layer and profiles of horizontal and vertical velocity through the vortex center. $\bar{u}_{inlet}$ marks the location of the inflow reference velocity and $\bar{u}_\infty$ the free stream velocity. Streamlines visualize the mean cavity flow. Data for profiles and streamlines are from the case 10m.

The different flow fields in the cavity for each case can be characterized by the mean volumetric flow rate of the main cavity vortex. The two dimensional flow rate ($\dot{V} \ [m^3/s/m]$) is derived by integrating the horizontal profile of mean vertical velocity (Fig. 4.14). The following procedure was used. The magnitude of the mean vertical velocity profile over a horizontal line from the left to the right cavity wall (cavity width $W_c$, Fig. 4.14):

$$\dot{V}(y) = \frac{1}{2} \int_0^{W_c} |\bar{u}_y(x, y)| \, dx$$  \hspace{1cm} (4.6)
The velocity at the left and the right wall was set to 0 m/s. An additional point was inserted at the center of the velocity profile where \( |\overline{u}_y| = 0 \), found through a linear interpolation. The flow rate (Eq. 4.6) was evaluated at multiple heights to find the maximum value \( \dot{V}_{\text{max}} \) (due to the discrete locations of the PIV pixels, a three-point parabolic fit was used to interpolate).

The cavity flow rate (Eq. 4.6) can easily be converted into an average bulk vortex velocity \( \overline{u}_{\text{vort}} \):

\[
\overline{u}_{\text{vort}} = \frac{2}{W_c} \dot{V}_{\text{max}}
\]  

(4.7)

Figure 4.15 shows the flow rate \( \dot{V}_{\text{max}} \) normalized by the case ThinBL for 30 different spire/barrier combinations. In most cases (24 out of 30), an added disturbance increases the mean volumetric cavity flow rate in the canyon over the case with no disturbance (ThinBL). The fastest flow rate is up to 50% higher, the slowest 20% lower. When comparing the flow rate to the geometry of the disturbance, a clear clustering with respect to the width-to-height ratio is observed. The narrow spires result in a higher flow rate than the wide spires. For neither the spire height nor the barrier height is a correlation to the flow rate apparent. Exceptional are the cases Bo.6 and B1.2 (only barriers, no spires), which exhibit the highest flow rate.

The flow rates depicted in Figure 4.15 can also be understood as if they would be scaled by the free stream velocity, since it is the same for all cases (\( \approx 3.14 \) m/s). An upstream scaling velocity \( \overline{u}_{\text{inlet}} \) (Fig. 4.14) near the wall \( (y = 5 \text{ mm}, \text{averaged between } x = -40 \text{ mm and } x = -20 \text{ mm}) \) was chosen to scale the mean vortex velocity (Eq. 4.7). The ratio \( \overline{u}_{\text{vort}} / \overline{u}_{\text{inlet}} \) can be interpreted as a factor of how much momentum is transported from the external flow to the cavity. The result, displayed in Figure 4.16, is opposite to the previous figure (Fig. 4.15). The wide spires show the largest velocity, whereas the narrow spires show the lowest velocity. A correlation of the velocity with the turbulence intensity of the approach flow is apparent. This is not surprising, as the rotation velocity of the cavity vortex can be understood as the amount of momentum being transferred by the flow aloft to the cavity. Since there exists a dividing streamline between the up- and downstream cavity edge, momentum transfer can only happen through shear stress (turbulence) and not through the mean flow velocity. However, the mean velocity of the flow represents the
Figure 4.15: Normalized average volumetric cavity flow rate, colored by spire width-to-height ratio. The error bars show the difference between the downward and upward flow rate.

Figure 4.17 shows the position of the mean cavity vortex center. The majority of cases are clustered near the horizontal centerline and slightly offset (by 4 mm) towards the right wall. This can be explained by a higher maximum velocity on the downward (right) side than the upward (left) side (assuming mass conservation and two dimensional flow). The higher maximum velocity on the right side arises through flow entrainment at the downstream cavity edge. Most wide spire cases show different vortex center locations. The vortex generated by the wide spires might not have a defined stationary center. The vortex is disturbed by large three dimensional flow structures, resulting in lateral mean velocity gradients (as visible in Fig. 4.11).
Figure 4.16: Ratio of mean vortex velocity to mean upstream velocity, colored by spire width-to-height ratio. The markers show the degree of (upstream) turbulent intensity.

Figure 4.17: Position of the mean vortex center. The dashed lines are the cavity centerlines.
The shear layer can be characterized by its TKE production. Three distinct shapes are found when comparing the normalized TKE production fields ($P_{TKE}/P_{TKE,max}$, Fig. 4.18). The majority of the cases feature a thin elongated ellipse, located in the first half of the shear layer. Production in the second half can only be observed for the case ThinBL. A short production area is visible for the cases with a high degree of turbulence.

One case for each type of production has been selected, 5m, ThinBL and 15w. This section describes these cases in more detail. For the case ThinBL, a boundary layer was developed naturally by the inlet fetch. For the two cases 5m and 15w, additional turbulence was generated by spires with height 5 cm and 15 cm respectively. The width to height ratio of the 15 cm spire is double the value of the 5 cm spire (Tab. 4.1). The case 5m was selected due to the well mixed wakes at the cavity location, as was observed in the Hot Wire (HW) measurements. The case 15w was selected due to the high of turbulence and the large width of the spire, resulting in large scale turbulence. To not increase the complexity of the cases, no barriers are used.

### 4.6.1 Inlet Flow and Cavity Flow Field

Inlet boundary layer profiles upstream of the cavity are presented in Figure 4.19, showing the resulting values for the streamwise velocity $u_x$, TKE and shear stress $u'_1 u'_2$. The profiles are computed from the time-averaged PIV measurements and horizontally averaged between $x = -40 \text{ mm}$ and $x = -20 \text{ mm}$. The origin of PIV coordinate system is on the upstream cavity edge (Fig. 4.20). The boundary layer thickness corresponds with the spire height. A clear difference in the magnitude of TKE characterizes the degree of turbulence for the three cases. The case ThinBL shows virtually no TKE, and the small amount visible close to the wall is accounted to reflection artifacts. The moderately turbulent case 5m shows almost constant shear stress up to $y = 30 \text{ mm}$. The velocity profile for the case 15w shows that this case does not represent a developed boundary layer. The hot wire measurements (Fig. 4.9 and 4.10) confirm that the flow is still developing at the cavity location. Additionally, Figure 4.19 also shows an estimate of the streamwise integral length scale for
Figure 4.18: Normalized production of TKE ($\frac{P_{TKE}}{P_{TKE,max}}$) for the shear layer of the measured cases, sorted in columns by the normalized vortex velocity.
Figure 4.19: Inlet flow profiles averaged between $x=-40$ mm and $x=-20$ mm, showing mean velocity, turbulent kinetic energy, shear stress and integral length scale. Markers are shown for every 5th datapoint.

$\bar{u}_x$. $L_{1,x}$ is relatively constant in the lower 50 mm of the boundary layer and averages to 42 mm for the $15w$ case. This is about 2.6 times larger compared to the case $5m$ (16 mm). The length scale is still smaller than the cavity length (100 mm).

The cavity flow fields are presented in Fig. 4.20. All three cases show a clockwise rotating main vortex within the cavity and a horizontal flow above the cavity. Small corner vortices are formed in the bottom corners and in the top upstream corner. The cavity vortex is contained within the cavity by a dividing streamline, starting at the detachment point at the upstream edge and ending in the stagnation point at the downstream edge. The dividing streamline is horizontal and flat, except for the case $15w$, where the streamline slightly curves above the top. The velocity snapshot shows the structure of the turbulent flow in the cavity. For the case ThinBL and $5m$, a jet of higher momentum flow on the downstream wall is clearly visible (also in Fig. 4.21b).

Profiles for the main flow statistics are presented on lines through the cavity center ($x_c = 0.05$ m, $y_c = -0.05$ m) in Figure 4.21 and 4.22. The
vertical profiles in Figure 4.21a clearly show large gradients of $\overline{u}_x$ for the cases ThinBL and $5m$, with an inflection point just above the cavity top. An inflection point is a necessary condition for flow instability. The location of the inflection point corresponds to the point of maximum TKE and shear stress. The case $15w$ shows a shallow gradient and no clear peak of either TKE or shear stress. The horizontal profiles of mean vertical velocity $\overline{u}_y$ through the cavity center presented in Figure 4.21b provide information about the main cavity vortex. For both the cases ThinBL and $5m$, a jet of higher velocity is visible on the right side wall. Consequently, due to conservation of mass, the vortex center (intersection of $\overline{u}_y$ with $x = 0.05$ m) is offset to the right. The case $15w$ shows a flatter velocity profile. From $\overline{u}_y$ it can also be seen, that the average rotation speed of the cavity vortex is the highest for the case $5m$, followed by the case ThinBL and $15w$. All cases show an increased peak-like TKE and shear stress on the right wall. The case ThinBL shows a narrow and shear peak of shear stress, but the lowest TKE. The shear stress of case $15w$ shows only a small broad peak, however this case features the highest TKE.

The profiles of the three diagonal components of the Reynolds stress tensor ($\overline{u_1'^2}$, $\overline{u_2'^2}$ and $\overline{u_3'^2}$) are shown in Figure 4.22a and 4.22b. For all
three cases it can be observed that inside the cavity the magnitude of the
three components is about equal, meaning each direction contributes
equally to $TKE$ (turbulence is three dimensional). Figure 4.22a shows the
components of $TKE$ in the shear layer. $TKE$ of the case ThinBL is mostly
composed of streamwise fluctuations. This leads to the assumption, that
the shear layer turbulence for this case is not three dimensional. The
case $5m$ also shows the highest contribution to $TKE$ coming from the
streamwise fluctuations $u_1'^2$, however this case also shows significant
contributions from $u_2'^2$ and $u_3'^2$. The case $15w$ shows almost equal values
for all three components (with only $u_2'^2$ being slightly lower), indicating
three dimensional turbulence. Figure 4.22b shows horizontal profiles
through the cavity center. Notable is the high contribution of the lateral
component $u_3'^2$ to the $TKE$ on the right wall, especially for the case $15w$.
This means that the turbulent structures entrained develop at the wall
into three-dimensional turbulence, independent of the two-dimensionality
of the downstream cavity edge or the shear layer vortices.

4.6.2 Shear Layer

Besides the flow profiles, field plots of the turbulent statistics (e.g. $TKE$
and shear stress) show the connection of the cavity flow to the shear layer
(Fig. 4.23). All three cases show some $TKE$ inside the cavity, meaning
that the cavity flow is always turbulent. The case $15w$ shows the highest
$TKE$ both inside the cavity and in the boundary layer above. Compared
to the other cases, $TKE$ is also the most uniformly distributed in the
cavity. The distribution of $TKE$ at the cavity top (around $y = 0 mm$)
is significantly different for each case. The case ThinBL shows a large
amount of $TKE$ towards the downstream edge, $5m$ features high $TKE$
over the entire top, linearly growing in height and the case $15w$ shows a vertical
gradient of $TKE$ from the cavity to the boundary layer. This gradient is
quite pronounced at the upstream edge and low at the downstream edge.
Especially for the case $5m$ and $15w$, $TKE$ is also visible at the downstream
cavity wall. The production of $TKE$, $P_{TKE}$, shows a similar picture, with
most of the production concentrated in the second half along the cavity
top for the case ThinBL, but opposite for the case $5m$. The case $15w$ shows
only weak production at the upstream edge.
Figure 4.21: Profiles of mean velocity, TKE and shear stress $u_1'u_2'$ for the cases ThinBL, 5m and 15w through the cavity center for a) a vertical line at $x = 0.05$ m and b) a horizontal line at $y = -0.05$ m
Figure 4.22: Profiles of Reynolds Stress components $\overline{u_1'^2}$, $\overline{u_2'^2}$ and $\overline{u_3'^2}$ for the cases ThinBL, 5m and 15w through the cavity center for a) a vertical line at $x = 0.05 \text{ m}$ and b) a horizontal line at $y = -0.05 \text{ m}$
The case $5m$ fits the description of defining a turbulent shear layer through a clearly increased shear stress $-u'_1 u'_2$ along the cavity top accompanied by production of TKE. Also the case ThinBL shows the existence of a shear layer, even if less homogeneous over the full cavity length. For the case $15w$, there is neither a clear increase of shear stress nor an increase in $P_{\text{TKE}}$.

The power spectral density $\Phi$ of the horizontal and vertical velocity signal, $u_x(t)$ and $u_y(t)$ has been determined from the time resolved PIV measurements. The results are shown in Figure 4.24 and 4.25, for the horizontal and vertical velocity respectively, at 5 locations along the cavity top (as indicated in Fig. 4.23 by the diamonds). Further explanations to the velocity spectra are made in the next section.

The time resolved PIV measurements, together with the measured time averaged flow field, allow for plotting the spatio-temporal evolution of fluctuating velocity components along a specific line. This was done for a line along the cavity top for each case and is presented in Figure 4.26a and 4.26b, for $u'_x(t)$ and $u'_y(t)$ respectively. From the 5 seconds of acquired data, only the first 1.3 seconds are shown here. It is immediately apparent that the dynamics of the three cases are fundamentally different. These will be addressed in the next section in detail.

4.6.3 Discussion

This section highlights and discusses some interesting and unique properties of each of the three cavity flows. The cavity flow field statistics, the spectral properties of the shear layer and the spatio-temporal evolution of the flow have been presented in Section 4.6.1 and 4.6.2. Recall that for each case, the cavity geometry and the wind tunnel free stream velocity was kept constant and only the upstream turbulence generation was changed.

4.6.3.1 ThinBL

For the case ThinBL only the inlet fetch has been used to generate a boundary layer without adding turbulence generating spires. The cavity inlet profile (Fig. 4.19) therefore shows a thin boundary layer and no visible TKE. A further investigation into the shape of the boundary profile (Fig. 4.27) suggests that the profile is laminar and not turbulent. This is also supported by the Reynolds number $Re_x \approx 165'000$ (at a distance
Figure 4.23: Turbulent kinetic energy (TKE), Production of TKE and shear stress for the three cases ThinBL (top), 5m (middle) and 15w (bottom). White diamond markers show the location of point probes used for spectral analysis.
Figure 4.24: Power spectral density $\Phi$ of the horizontal (streamwise) velocity component $u_x$ for three cases ThinBL (left), $5m$ (middle) and $15w$ (right), evaluated at different horizontal locations along the cavity top.

Figure 4.25: Power spectral density $\Phi$ of the vertical velocity component $u_y$ for three cases ThinBL (left), $5m$ (middle) and $15w$ (right), evaluated at different horizontal locations along the cavity top.
Figure 4.26: Space-time plots of fluctuating velocity $u_x'$ (a) and $u_y'$ (b) in the shear layer at $y = 2\ mm$. 
Figure 4.27: Inlet profile of case ThinBL in comparison to the Blasius solution for a laminar boundary layer and the power law of a turbulent boundary layer, normalized by the boundary layer thickness and the free stream velocity.

$x = 0.86 \, \text{m}$ behind the inlet flap) which is below the critical $Re_x$ for a flat boundary layer ($Re_{xcrit} \approx 500'000$). The flow inside the cavity however is turbulent, as can be seen from the level of TKE measured by the averaged flow statistics (Fig. 4.21). A particular strong area of TKE and shear stress is observed in the last half of the shear layer (Fig. 4.23). This coincides with a very large production of TKE. The spectra of horizontal (Fig. 4.24) and vertical velocity (Fig. 4.25) reveal clear peaks, with a fundamental frequency and several harmonics. The power spectral density (and also number of harmonics) grows with downstream distance, consistent with the growth of TKE. It should be noted that the harmonics are magnitudes smaller than the fundamental frequency. Animations of the velocity fields show the initial wavelike disturbances to roll-up into large vortices that are mostly two dimensional with only small, occasional out-of-plane velocity. The shear layer, with its distinct vortex shedding, is analyzed in more detail.

With a momentum thickness of $\theta = 1.18 \, \text{mm}$, the ratio of the cavity width to the momentum thickness ($W_c / \theta$) is 85. This value falls into the
range between 80 and 160 where Gharib and Roshko (1987) measured self-sustained shear layer oscillations for a laminar upstream boundary layer. Self-sustained shear layer oscillations occur, when the vortex shedding is self-oscillating, a mode of \( m = 2 \) according to the semi-empirical model proposed by Rossiter and Britain (1964) is expected (Eq. 4.8):

\[
f = \frac{\overline{u}_c}{W_c (m - \gamma)}
\]

where \( f \) is the fundamental frequency, \( \overline{u}_c \) being the convective velocity and \( \gamma \) an empirical parameter given as 0.25. To get \( f \) and \( \overline{u}_c \) from PIV, the equation of a spatially growing wave (Eq. 4.9) was fitted to the space-time data of vertical velocity at \( y = 2 \ mm \) (Fig. 4.28b) using a Nelder-Mead donwhill simplex algorithm. 2000 PIV snapshots were used (corresponding to about 100 periods).

\[
u'_{y}(x,t) = a + be^{\beta x} \cos(\kappa x - \omega t)
\]

From the fitted wavenumber \( k \) and angular frequency \( \omega \), the frequency is calculated as \( f = \omega / (2\pi) \) and the convective velocity as \( \overline{u}_c = \omega / k \). This method of simultaneously extracting the frequency and the wavelength has proved to be far more robust than a combination of spectral analysis for the frequency and lagged two-point auto-correlation for the convective velocity. The result from the fitting are \( \overline{u}_c = 1.78 \ m/s \) and \( f = 31.5 \ Hz \). Together with Equation 4.8, the cavity oscillation mode results in \( m=2.04 \), relating the shear layer vortex shedding to self-sustained oscillations.

In addition to the fundamental frequency, the shear layer also exhibits low frequency disturbances. These disturbances are clearly visible in Figure 4.26a as large areas of increased or decreased horizontal velocity. Since the space-time representation of the horizontal velocity fluctuations is highly sensitive to a vertical displacement of the shear layer, we assume this is due to a low frequency flapping. To assess that this is not caused by external disturbances, horizontal space-time plots at different heights are extracted (Figure 4.28). At \( y = 8 \ mm \) above the cavity top, no disturbances can be observed upstream of the cavity, indicating that these disturbances are not present in the approach flow. When comparing the vertical fluctuations at \( y = -4 \ mm \) and the horizontal fluctuations at \( y = 2 \ mm \), a correlation can be observed, consistent with a low frequency flapping. This low frequency flapping of the shear layer has been observed in previous measurements and was studied in detail by Liu and Katz (2013).
Figure 4.28: Space-time plots of fluctuating velocity $u'_x$ (a) and $u'_y$ (b) in the shear layer at three locations ($y=8$ mm, $y=2$ mm, $y=-4$ mm) for the case ThinBL.
Figure 4.29: Inlet profile of case $5m$ compared to a fit with a power law, normalized by the boundary layer thickness and the free stream velocity.

4.6.3.2 $5m$

By placing 5 cm tall spires ($H_s = 0.5H_c$) upstream of the cavity, a turbulent boundary layer is generated with a moderate level of turbulence intensity (2%) and a momentum thickness of $\theta = 7.76 \, \text{mm}$. The boundary layer is composed of a decaying spire wake flow and a growing internal boundary layer generated by the inlet fetch. This makes the observations specific to the location of the cavity (distance away from the spire). However, the cavity is located sufficiently far downstream of the spires that the geometrical pattern of the spires is not visible in the average flow field (Fig. 4.11). In addition, the flow profile is close to a power law profile (Fig. 4.29). Together with the TKE and shear stress profiles reported in Figure 4.19, the $5m$ spire setup is capable of simulating a turbulent boundary layer.

The analysis of the average flow field shows a clear increase of TKE in the shear layer (Fig. 4.23), and the thickness increases linearly with downstream distance. The production of TKE is concentrated in the first half of the shear layer, with a maximum at $x = 20 \, \text{mm}$. To elucidate this peak, the contribution of each term in the TKE production equation
Figure 4.30: Spatial evolution of $P_{TKE}$ along the cavity top ($y=2$ mm) for the case 5$m$, together with the dominant term and its components. Each value is normalized by its maximum.

(Eq. 3.19) is compared. The term $\overline{u_1' u_2'} \frac{\partial \bar{u}_1}{\partial y}$ is found to be dominant and is shown in Figure 4.30. The high velocity gradient at the beginning of the shear layer and the rising shear stress culminate to a peak and then fall to an almost constant level up to where the downstream cavity edge becomes influential. The characteristics are similar to what can be found for a backward-facing step (Kostas et al. 2002). A backward-facing step can be interpreted as a cavity without a downstream wall. The flow for a backward-facing step also separates at the upstream edge, creating a large vortex in the wake area. Similarly, a shear layer exists between the low speed wake vortex and the higher speed flow aloft. The main difference is that the flow for the backward-facing step reattaches to the bottom wall further downstream, whereas the cavity shear layer impinges on the downstream edge. Therefore, the shear layer of a backward-step cannot show self-sustained oscillations and the main mechanism that produces turbulent vortices is the Kelvin-Helmholtz instability. The similarity of the 3.19 profile therefore indicates that this cavity shear layer is also dominated by the Kelvin-Helmholtz instability.

The shear layer velocity spectra (shown in Fig. 4.24 and 4.25) show an increase of power spectral density $\Phi$ with increasing distance, most notably for the vertical velocity between $x = 10$ mm and $x = 30$ mm and less pronounced up to $x = 70$ mm. Compared to the case ThinBL, no clear
peak that would correspond to a fundamental frequency is visible. To show the growth in spectral power, noise removal had to be applied by using Welch’s Method (Welch 1967) when generating the spectra. This however also removes sharp peaks. For single point spectra with no smoothing, it is difficult to distinguish peaks from noise, therefore a spatial spectrogram was computed. From the spatial persistence, it is possible to identify spectral peaks that originate from shear layer flow structures. Figure 4.31 shows clearly that characteristic frequencies exist. The largest peak is observed at 30.81 Hz. The associated $\Phi$ grows until the position reaches about 80 mm downstream. After that, the peak at 29.1 Hz becomes dominant. It appears, that the turbulent kinetic energy is transferred between the frequencies. This could be due to a vortex pairing and breakup, and/or interaction with boundary layer coherent structures.

An example of such complex vortex interaction, pairing and breakup is shown in the snapshots in Figure 4.32. In addition, Fig. 4.32 visualizes the out-of-plane velocity component $u_z$, showing that the vortices in the shear layer, other than in the case ThinBL, are not predominantly two-dimensional, but interact with the three-dimensional structures of the boundary layer. Besides the constant shedding of new coherent structures in the shear layer, and their interaction with each other and the external flow, no other repeating pattern can be observed when looking at more snapshots. This lack of a single dominant periodicity can also be seen in the space-time plots (Fig. 4.26).

The momentum thickness was determined by integrating the boundary layer profile. With a momentum thickness of $\theta = 7.76 \text{ mm}$ the ratio $L/\theta$ is equal to 13. This is well below the range where self-sustained oscillations are expected to occur. This, together with the absence of a clear fundamental frequency, means that the main mechanism for vortex formation could be the natural Kelvin-Helmholtz instability of the shear layer, but disturbed by the turbulent structures of the external flow.

4.6.3.3 15\text{w}

The third case features a turbulent upstream boundary layer with 15% turbulence intensity, generated by 15cm tall spires ($H_s = 1.5H_c$), with a width of 4.6 cm at the base. The hot wire measurements (Fig. 4.11) show a large wake behind the spires. The mean flow velocity measurements suggests that the spire wakes of multiple neighbouring spires join
4.6 Shear Layer Analysis

Figure 4.31: Shear layer power spectral density $\Phi$ of $u_y$ for the case 5m showing a) the spatial evolution in the shear layer and b) the magnitude of the spatial average.

Figure 4.32: Sequential snapshots of the case 5m, showing instantaneous fluctuation velocity (vectors), out of plane velocity $u_z$ (color) and vortex cores obtained from $Q^{2D}$ (black).
together to form a larger, stationary wake. This is presumably due to the large blockage ($\approx 30\%$) of the spires, making them act in parts like a barrier where flow is pushed over the top and not predominantly act like turbulence generators.

The location of the cavity is downstream of the main wake zone of the spires. With the spire wake reattachment length $x_r$ measuring around 0.3 m, this places the cavity roughly twice $x_r$ downstream. This results in a cavity inlet flow with distinctive events of large disturbance. This can be seen especially well in Figure 4.26b, where large events of negative vertical velocity are observed, for example at $t=0.8$ s.

The time resolved data acquired with the PIV system spans a duration of $T_{\text{meas}} = 5.26$ s, or roughly 120 integral time scales (with $T_{1,x} = 0.04$ s). Even though the spire flow is statistically stationary, this assumption is not valid over the duration $T_{\text{meas}}$. Therefore, statistical analysis that uses the underlying assumption of stationarity does not represent the flow well. If we consider an event to be a bulk motion of fluid into or out of the cavity, a high momentum but infrequent event would not appear in e.g. the autocorrelation function (used to determine the integral time scale), or the spectral analysis shown in Figure 4.24 and 4.25. In order to find and classify events, the quadrant method has been applied. However, instead of using uncorrelated PIV snapshots to compute the histogram of the four quadrants, the shear stress $\overline{u_1' u_2'}$ was computed from spatio-temporal velocity fluctuations (Fig. 4.26), spatially averaged over the cavity top and subsequently decomposed into the four quadrants. The resulting time evolution is shown in Figure 4.33.

It is immediately apparent that sweep events dominate the shear layer. Within this measurement period, two very strong sweep events are detected, around $t = 0.8$ s and $t = 3.4$ s. Ejection events also occur, and are similar in magnitude than the smaller, more numerous sweep events. Further insights into these events can be gained by plotting the spatio-temporal evolution of sweep/ejections overlaid with the vortex criteria $Q$ in the shear layer. In the left panel in Figure 4.34, we observe that $Q$ is usually associated with sweep. The slope of structures relates to the velocity $\overline{u_c}$ at which these structures are convected through the shear layer. We look at four events (as defined in Fig. 4.34). The sweep events A and D show a higher $\overline{u_c}$ (seen in the steeper slope) than event C. The two events A and B seem to be associated with large scale, high velocity vortical structures in the boundary layer, presumably generated by the interaction
of the spire wake with the high velocity free stream flow. Event C in contrast features smaller structures, resembling more the structures of a turbulent boundary layer. Whilst the sweep in event A and C is mostly convected over the cavity, event D shows a penetration of high momentum fluid into the cavity. This area of high momentum also contains numerous small coherent structures that are transported into the cavity. Additionally, new vortical structures form at the interface of the sweep and the low momentum cavity fluid.

A distinct mechanism of vortex formation can be observed for all sweep events (A, C and D): high momentum fluid passes over the upstream corner of the cavity, separates at the edge and due to the high velocity gradient sheds a vortex with the axis parallel to the cavity edge. This is supported by the correlation of a sweep at the leading edge, and the subsequently detected vortex (left panel in Fig. 4.34). This vortex shedding mechanism shows up as a weak production of TKE, as visible in

Figure 4.33: Temporal evolution of the spatially averaged shear stress $\langle u_1' u_2' \rangle$ along the cavity top at $y = 2\, mm$. Colored lines show the contribution of the 4 quadrants, and the colored areas show the difference in magnitude of sweep and ejection.
Figure 4.34: Space-time representation (left) and snapshots (right) of sweep (red) and ejection (blue) events in the shear layer at $y = 2\, \text{mm}$, overlaid by the vortex cores obtained from $Q^{2D}$ (black). Vectors are instantaneous fluctuation velocity. The snapshots represent 4 selected events, from top to bottom: sweep (A), ejection (B), sweep (C), sweep (D).

Figure 4.23. It is believed, that this is also responsible for the peaks in the power spectral density at $x=10\, \text{mm}$ and $x=30\, \text{mm}$ (Fig. 4.24 and 4.25).
4.7 DISCUSSION AND CONCLUSION

Time resolved and time averaged stereo-PIV measurements have been performed on a 2D unit aspect ratio shear-driven cavity. Cavity flow experiments are usually conducted for a specific type of inflow condition. In this experiment, a split-floor design was used in combination with or without turbulence generating spires to successfully generate different inflow boundary layers, from laminar to highly turbulent.

The experimental setup was capable of producing two typical cavity flows. The case with the thin boundary layer shows the expected self-sustained oscillations at a distinct frequency. A thicker, more turbulent boundary layer shows classical shear layer growth and vortex shedding through Kelvin-Helmholtz type instabilities. Even though self-sustained oscillations can occur when the upstream boundary layer is turbulent and thicker, our measurements showed no clear indications that resonance phenomena are present. The third case features highly turbulent and intermittent inflow conditions, for which no comparable measurements exist in current literature. The case can be interpreted as a street canyon located in the wake of a larger upstream obstacle (building or topography). Instead of vortex formation in the shear layer, as observed with the moderately turbulent inflow, vortex shedding can be observed at the upstream corner, due to upstream edge flow detachment caused by strong sweep events. The main observations made from the three specific boundary layers are summarized as follows:

- The mean flow topology (streamlines) is similar for each case, showing a large main vortex, two corner vortices at the floor and a vortex at the upstream corner. This suggests that for this experiment the cavity geometry (aspect ratio) determines the vortex pattern.

- A clear shear layer, defined through shear stress and production of TKE, was observed when the upstream turbulence was not dominating.

- Coherent structures at the cavity top are produced in each case, however the production location and mechanism is significantly different (self-sustained oscillations versus Kelvin-Helmholtz vortices versus upstream edge vortex shedding)
• Not only relying on statistical analysis based on mean flow quantities and spectral analysis revealed transient events.

The measurements have demonstrated the significant differences of the cavity flow when large scale turbulence and disturbances exist in the upstream boundary layer. It is therefore expected, that the exchange of scalars such as pollutant and heat in and out of the cavity is governed significantly by these large scale events, instead of the shear layer dynamics.

The observed, distinct events of flow entrainment (sweeps) and flushing (ejections), pose difficulties for RANS modellers since these flows are not predominantly described by classical shear layer statistics. Time resolved PIV in combination with scalar transport, such as PIV-Laser Induced Fluorescence (LIF) or Quantitative Light Sheet (QLS) enables measurements of flushing events and computing ensemble averages. The main challenge for time resolved numerical simulations (e.g. Large Eddy Simulation) is to efficiently provide highly turbulent and intermittent inflow conditions, such as observed in these measurements.

The boundary layer just upstream of the cavity, the cavity shear layer and the cavity flow are all measured in a single frame. Therefore, the design of the measurement setup provides data with large spatial extent and with sufficiently high spatial and temporal resolution. Together with the distinct flow phenomena present under the different inflow conditions (laminar, moderately turbulent and highly turbulent) allow the verification of cavity flow simulations performed using Large Eddy Simulation (LES). However, due to the limitations of the PIV system, only a limited number of snapshots can be acquired. This is especially important when computing statistics from time resolved measurements, such as the ACF and the subsequent determination of the integral time scale (defined as the area under the ACF). The statistical uncertainty of the autocorrelation has been assessed using synthetically generated velocity signals and it was found that the conventional method of direct integration of the ACF, under the conditions found in the experiment, is not feasible. Curve-fitting has been identified as a viable method. The measurement setup was found capable of generating datasets of high quality for the development and validation of turbulent inflow conditions for LES.
This chapter describes the computational model used to simulate the air flow for urban configurations. The model is based on unsteady Computational Fluid Dynamics (CFD) using the Large Eddy Simulation (LES) approach. LES is computationally demanding but has the potential to give physically realistic results.

Section 5.1 describes the scope of the model and highlights the possibilities. The scale and resulting computational domain for which the model is developed is presented in Section 5.2. Section 5.3 describes the details of the LES model. The code used is described in Section 5.4.

5.1 Scope

The scope of the urban air model is limited to resolving the instantaneous wind field and the transport of passive scalars. This scope is sufficient to investigate the feasibility of the LES approach for urban flows, and identify the main challenges unique to LES. This scope allows for comparison to time resolved Particle Image Velocimetry (PIV) wind tunnel measurements (isothermal flow). The transport of passive scalars allows for visualizing the flow by tracing the movement of air passing through defined locations and investigating the effects of the model parameters on the dispersion of scalars. The model focuses on the transport within the air domain, meaning for scalars released directly into the urban air or already present (e.g. release from air conditioning vents, or pollutants), rather than the transfer through walls (e.g. convective heat transfer). The scope of the model can be extended to include non-isothermal effects (buoyancy), heat transfer by radiation and convection and chemical processes in the non-idealized air (reacting pollutants, moisture).
5.2 Scale and Computational Domain

For a LES model to be applicable to urban, local scale flows, it needs to resolve buildings with sufficient spatial resolution. With limited computational resources, high resolution can only be achieved by limiting the spatial extent of the domain. The flow field around a single building of interest embedded in an urban environment can only be predicted accurately, if the local wind conditions are imposed correctly. For a given wind direction, the mean flow field (short term) depends on the surrounding buildings, as they can create channel and wake flows. The turbulent flow field depends on the upstream buildings (and other obstructions, such as vegetation) and the turbulent structures present in the lower part of the Atmospheric Boundary Layer (ABL).

In order to limit the extent of the computational domain, turbulence that is not generated by buildings included in the computational domain, has to be provided by the inlet condition. From the existing approaches (Sec. 2.6.4), this model employs synthetic turbulence generation (Chapter 6).

5.3 Large Eddy Simulation (LES)

A Large Eddy Simulation (LES) solves the filtered Navier-Stokes equations numerically, resolving most of the Turbulent Kinetic Energy (TKE). The unresolved energy is modelled by a sub-grid model. The LES approach has specific requirements with regard to the discretization, computational grid, boundary conditions, flow initialization and data sampling.

5.3.1 Filtered Navier-Stokes Equations

The differential form of the Navier-Stokes set of equations for an isothermal incompressible fluid in a Cartesian coordinate system are the continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (5.1)

and the momentum equation:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2}$$  \hspace{1cm} (5.2)
where $\rho$ is the fluid density and $\nu$ the kinematic viscosity.

By applying a filter kernel through a convolution, the filtered velocity field is obtained:

$$\tilde{u}(x,t) = \int G(r,x)u(x-r,t)dr.$$  \hspace{1cm} (5.3)

The velocity field can therefore be decomposed into a filtered and a residual part:

$$u(x,t) = \tilde{u}(x,t) - u'(x,t).$$  \hspace{1cm} (5.4)

These two equations allow us to write the filtered Navier-Stokes equation as

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0$$  \hspace{1cm} (5.5)

$$\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_j^2} - \frac{\partial \tau_{ij}}{\partial x_j}$$  \hspace{1cm} (5.6)

with the tensor

$$\tau_{ij} = \bar{u}_i \bar{u}_j - \tilde{u}_i \tilde{u}_j$$  \hspace{1cm} (5.7)

being the residual stress tensor, or sub-grid stress (SGS) tensor. The SGS tensor arises through the filtering of the non-linear convection term $\partial(u_i u_j)/\partial x_j$. The residual stress is posing a closure-problem and has to be modeled by an appropriate sub-grid scale model.

5.3.2 SGS Model

Due to the turbulence closure problem, the unknown subgrid stress tensor $\tau_{ij}$ in equation 5.6 has to be modelled. A commonly used class of SGS models are based on the eddy viscosity hypothesis, where the residual stress tensor is related to the resolved strain rate tensor $S_{ij}$ through the subgrid viscosity $\nu_{sgs}$:

$$\tau_{ij} = \frac{2}{3} \kappa_{sgs} \delta_{ij} - 2\nu_{sgs} S_{ij}$$  \hspace{1cm} (5.8)

with the strain rate tensor defined as

$$S_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)$$  \hspace{1cm} (5.9)
and $\kappa_{\text{sgs}}$ as the turbulent kinetic energy of the residual velocity field. This class of models are therefore called the eddy viscosity models.

The values for $\nu_{\text{sgs}}$ and $\kappa_{\text{sgs}}$ can either be algebraically computed (zero-equation models), as with the Smagorinsky model, or an additional transport equation for $\kappa_{\text{sgs}}$ can be derived (one-equation models).

A one-equation model is used. One-equation models have been developed by Schumann (1975) and Yoshizawa and Horiuti (1985). Similar to the RANS TKE equation (Eq. 3.18), a transport equation for the turbulent kinetic energy of the residual velocity ($\kappa_{\text{sgs}} = 1/2 (u'u')$) can be derived. The $\kappa_{\text{sgs}}$ transport equation reads:

$$\frac{\partial \kappa_{\text{sgs}}}{\partial t} = -\tilde{u}_i \frac{\partial \kappa_{\text{sgs}}}{\partial x_j} + 2 \nu_{\text{sgs}} S_{ij} \frac{k_{\text{sgs}}^{3/2}}{\Delta} + \frac{\partial}{\partial x_i} \left[ (\nu + \nu_{\text{sgs}}) \frac{\partial \kappa_{\text{sgs}}}{\partial x_i} \right]$$

with

$$\nu_{\text{sgs}} = C_k \Delta k_{\text{sgs}}^{1/2},$$

where $C_k$ and $C_\varepsilon$ are model constants. The symbol $\Delta$ is sometimes referred to as the filter width. However, for unstructured finite volume computations, the filtering operation is not explicitly formulated, but rather implicitly imposed by the grid. Therefore, $\Delta$ is not actually related to the width of the filter function $G(r, x)$, making $\Delta$ a (local) constant of the SGS model. A common choice is to relate it to the grid through the cell volume $V$ as $\Delta = \sqrt[3]{V}$.

The $\kappa_{\text{sgs}}$ transport equation (5.10) is similar to a scalar transport equation, with the additional second and third terms on the rhs being the production and dissipation of $\kappa_{\text{sgs}}$ respectively.

### 5.3.3 Scalar Transport Model

The transport of a conserved scalar quantity $\phi$ is modelled by the advection-diffusion equation:

$$\frac{\partial \phi}{\partial t} = -\tilde{u}_i \frac{\partial \phi}{\partial x_j} + \kappa_{\text{eff}} \frac{\partial^2 \phi}{\partial x_j^2} + S$$

(5.12)

The effective diffusion coefficient $\kappa_{\text{eff}}$ is defined as

$$\kappa_{\text{eff}} = \nu / Pr + \nu_{\text{sgs}} / Pr_t$$

(5.13)
where $v/Pr$ relates to the molecular diffusion and $v_{sgs}/Pr_t$ to the subgrid diffusion. For a generic passive scalar, for both $Pr$ and $Pr_t$, a value of 1.0 is assumed.

### 5.3.4 Discretization

For the spatial discretization of the convection term in the momentum equation, a second order central difference scheme would provide the least numerical dissipation. However, for urban flows, due to the relatively large cell size and therefore high Reynolds number, the scheme is prone to unphysical oscillations (Hirsch 2007), as the cell Reynolds number exceeds the criterion $Re > 2$ (Ferziger and Peric 2002) almost everywhere in the domain. Therefore, the Gamma scheme (Jasak et al. 1999) is used, with the scheme’s constant $\beta_m$ set to 0.5.

For the discretization of the convection of scalars and $\kappa_{sgs}$, a Total Variation Diminishing (TVD) scheme (Sweby 1984) is used, in order to ensure the boundedness of the solution.

For the time discretization, a second order implicit scheme is used.

### 5.3.5 Computational Grid

In contrast to a Reynolds-Averaged Navier-Stokes (RANS) computation, LES is inherently mesh dependent. Figure 5.1 sketches the spectrum of turbulence and shows the theoretical regions of resolved versus subgrid scales. Due to numerical dissipation from the discretization schemes and the solution approach, the resulting LES spectrum decays before the filter cutoff $1/\Delta$. For the finite volume approach with implicit filtering, the effective filter width $\Delta_{eff}$ is not known a-priori (Denaro 2011).

The important dynamic features of the flow must be captured by the resolved part of the spectrum that is unaffected by excessive dissipation. For practical applications, this makes the mesh size the defining parameter for the quality of a LES result and is case dependent.

### 5.3.6 Boundary Conditions

The boundary conditions for the velocity have been set to a no-slip condition for the ground and walls, symmetry at the top and zero gradient
Figure 5.1: Sketch of the turbulent spectrum and the spectrum resolved by LES.

at the outlet. For the pressure, the inlet, ground and walls have zero gradient conditions and the outlet is a zero pressure condition. Passive scalars $\phi$ can be released in two ways: by setting a fixed value at a wall boundary (e.g. $\phi = 1$) or by setting a source in a volume. For a flushing or ventilation study, the scalar in a certain volume can also be initially set to a fixed value $\phi = 1$.

For all fields, the lateral boundaries are periodic and therefore enforce laterally homogeneous flow.

The inlet velocity boundary condition for LES needs special treatment. For urban boundary layer flows, turbulent, time dependent velocity fluctuations have to be provided at the inlet. This model employs a synthetic turbulence approach and a separate chapter is dedicated to that topic (Chapter 6).

5.3.7 Initial Condition & Convergence

In order to extract meaningful results, the initial condition for a LES run should be a fully turbulent and statistically stationary flow field. This is achieved through an initialization phase. A second phase is used to sample results (Fig. 5.2).

We define convergence for a LES with regards to sampled statistics, and not the numerical iterations. For each timestep, the solution algorithm iterates until the required convergence of the momentum and pressure
equation is reached ($1 \times 10^{-5}$ and $1 \times 10^{-7}$). Convergence for the sampled statistics is defined in the same way as for PIV or hot wire wind tunnel measurements, by estimating the statistical uncertainty (as described in Section 4.3.3). The maximum number of samples to simulate is then determined by setting a desired maximum relative error for the mean velocity and the Reynolds Stresses.

The uncertainty of a LES result can therefore be reduced by:

- starting the data sampling from a statistically stationary initial state
- sampling for a sufficiently long time

The sampling of statistics has to be started from a statistically stationary flow field. This means that the mean flow velocity, the integral time scales and the Reynolds stresses do not change over time. It is however virtually impossible to determine from what exact point on a solution, evolving from an initial field, becomes statistically stationary. With a long initialization time $T_{\text{init}}$, stationarity can be shown. It is however the goal to minimize $T_{\text{init}}$, since this phase does not directly contribute to the sampling of results.

Generally, $T_{\text{init}}$ is estimated as a multiple of a flow-through time (e.g. Fröhlich et al. (2005)) defined by $T_{\text{init}} = n \times l_x / u_x$, where $u_x$ is a velocity and $l_x$ the length of the domain. For a case with wake flows, a wrong choice of $u_x$ can severely underestimate the flow through time. The number of flow-through times $n$, is therefore case dependent, making it unfeasible to propose generalized rules for the choice of $n$.

We propose a different approach to determine the $T_{\text{init}}$. The concentrations of passive scalars (released at specific locations) are sampled at the outlet of the domain. The scalars are continuously released in low velocity zones upstream (e.g. at the inlet near a wall or within a recirculation region). The longest time at which the first rise of concentration is detected at the outlet for each scalar can be regarded as a good estimation...
of the *flow-through* time. As soon as the sampled concentrations reach a steady state, the flow field can be considered statistically stationary. This is based on the assumption, that the flow field converges faster to a stationary state than the released scalars. To determine if a scalar is converged, an exponential function is fitted to the sampled concentration, allowing to set a threshold value to define convergence. Figure 5.3 shows an exemplary scalar during the initialization phase. This procedure can be evaluated periodically throughout the initialization run and the solver stopped when sufficient convergence is reached.

The initialization time can be further reduced by providing an initial (laminar) mean flow field that corresponds to the case geometry. Even for complex geometries, a RANS computation without a turbulence model on a coarse grid is already a sufficient initial field.
5.3 LARGE EDDY SIMULATION (LES)

5.3.8 Averaging and Runtime Sampling

Ideally, one would like to save as much data as generated by the simulation. In reality this is impractical, as this simple example shows: with a minimum result to store of the velocity vector (3 components) and the pressure (1 component), this equals 4 floating point numbers per grid cell. When stored as double precision 64-bit values (8 bytes), an exemplary mesh with 10 mio. grid cells requires 320 Megabytes per timestep. A typical run can have more than 500,000 timesteps, resulting in a minimum storage space of 160 Terabytes (note that solver-specific output, e.g. for restart purposes, can easily double or triple this amount). This is not only expensive in terms of storage space, but also highly impractical for post-processing purposes. A well set-up runtime processing can generate results during the simulation without storing all timesteps (either on disk or memory) and reduce post-processing time significantly.

A simple application of run-time processing is the computation of low-order statistics, such as the mean flow field and the Reynolds stress field. In this case, so called single-pass algorithms can be applied. The main idea is, that each timestep has to be read only once and not be kept in memory. For the mean, this is straightforward:

\[ \bar{X}_{i+1} = (\bar{X}_i \ast N_i + X_{i+1}) / N_{i+1}, \]  

and requires only storing the mean \( \bar{X}_i \) and the number of samples \( N_i \) from the previous timestep. The algorithm for the variance and co-variances, can be constructed in a similar way. The implementation is slightly more involved, since it is prone to numerical accuracy problems. A robust implementation as a one-pass algorithm is proposed by Welford (1962). A discussion on the stability and accuracy of different implementations can found in Chan et al. (2014).

In our simple example simulation mentioned above, if we would only be interested in obtaining first order statistics, the final storage space reduces to 3 doubles/cell for the mean flow velocity, 6 doubles/cell for the Reynolds stress tensor and 2 doubles/cell for the mean pressure and pressure variance. This is a total of 880 Megabytes instead of 160 Terabytes. This example highlights the need for runtime processing. For any other quantities of interest it must be carefully evaluated whether the full sampling frequency or sampling volume is needed. Often, point probes, volume integrals or 2D slices are sufficient.
The scalar covariance can be obtained using
\[
    \overline{u_i^' \phi^'} = \overline{u_i \phi} - \overline{u_i} \overline{\phi},
\]  
and only requires one to additionally sample and runtime average the product \( u_i \phi \). With Eq. (5.15), the final covariance is computed in post-processing (for the last timestep).

5.4 IMPLEMENTATION

The urban air model is implemented using the open source c++ library \textit{OpenFOAM} (OpenFOAM Foundation 2015). The \textit{OpenFOAM} library is a general purpose finite volume c++ code, that allows to efficiently implement and solve partial differential equations (Weller et al. 1998). \textit{OpenFOAM} features good scalability on High Performance Computing (HPC) infrastructure, enabling large mesh sizes.

The implemented \textit{LES} solver is based on the \textit{PISO} solution procedure (Issa 1986), and solves the filtered Navier-Stokes equations (Eq. 5.5 and 5.6), with the one equation SGS model (Eq. 5.10). In addition, the solver can handle any number of passive scalars (Eq. 5.12). The developed solver \textit{pisoScalarSourceFoam230} is available online (Immer and Vonlanthen 2015a), together with tools for runtime sampling and post-processing.

5.5 CONCLUSION

This chapter presented the computational approach used in this thesis to simulate urban flows. The \textit{LES} method was described and a procedure was presented that ensures a good quality of the acquired statistical results. The procedure determines the minimum initialization time for the simulated flow field by using passive tracers. The turbulent inlet boundary condition used in the model is described and further developed in the next chapter.
The principal goal of an inflow Boundary Condition (BC) for a Large Eddy Simulation (LES) is to provide a turbulent fluctuating velocity field. As this is required at every timestep, the method has to be computationally efficient.

This chapter presents an evaluation of existing methods and motivates the choice of the filtered noise method. The filtered noise method (Klein et al. 2003) is a synthetic turbulence generator, based on the digital filtering of random fields. The filtering is implemented according to the desired statistics of the generated flow field, i.e. mean flow velocity, Reynolds stress tensor and integral time- and length scales.

The length scales are imposed through a filtering of random fields, for which a new formal way is derived to obtain the coefficients. An efficient implementation for the flow code OpenFOAM was created, achieved through parallelization of the filtering operation.

In order to show the influence of the choice of different scales and filter functions (parameters of the method) on the generated flow, simulations were performed on a turbulent boundary layer. The inflow statistics were derived from time resolved stereo-Particle Image Velocimetry (PIV) measurements.

Section 6.1 evaluates different methods and motivates the choice of the filtered noise method, described in detail in Section 6.2. Section 6.3 presents a new formal way to obtain the methods parameters and details the implementation. The results of the boundary layer simulations are presented in Section 6.4 for the flow field and in Section 6.5 for the turbulent scalar, followed by conclusions in Section 6.6.

6.1 EVALUATION OF METHODS

Different existing methods to provide turbulent inflow conditions for LES were presented in Chapter 2.6.4. These included laminar-turbulent transition, cyclic main domain, cyclic driver domain and artificial turbulence
Evaluation criteria are computational overhead (efficiency), control over the generated turbulent flow, range of applicability and physically correct turbulent structures. The results of the evaluation are presented in Table 6.1.

All methods that need additional simulation time or a larger domain size create a computational overhead. These are computational resources that are only used to generate inflow conditions and are therefore not spent on the actual study. For the laminar-turbulent transition, this is in the form of additional computational cells. For the cyclic method, this is mostly computational time spent on the convergence of a statistically stationary state of the initial flow field. This also counts for the driver domain method. This method also has an overhead due to a higher cell count, as the driver domain is run in parallel to the main simulation. Turbulent inflow generators can be implemented efficiently and result in a very low computational overhead.

Control is evaluated based on the ability of the method to provide a turbulent boundary layer with a specified mean velocity profile, Reynolds stress tensor and integral length scales. This is especially important for LES validation with wind tunnel measurements. Transition, cyclic and driver domain based methods all depend on the spatial evolution of the flow adjusting to the surface roughness conditions. This can be controlled by changing the surface, e.g. by adding different blocks or obstacles. This however is an indirect form of control. A specific combination of mean flow velocity, Reynolds stress and length scale might be difficult to achieve as only one parameter (the surface geometry) can be influenced. This makes this a trial-and-error method, similar to the simulation of an Atmospheric Boundary Layer (ABL) in the wind tunnel where different
combinations of spires, barriers and roughness obstacles are iterated until
the desired ABL profile is reached. Due to the high computational cost of
LES this procedure is highly impractical. Inflow generators on the other
hand directly impose the desired flow statistics.

To further evaluate applicability, the wind tunnel setup from Chapter
4 is considered and presented in a simplified manner in Figure 6.1. To
simulate the cavity flow under the same conditions as found in the
experiment, the turbulent inflow at the cavity must match the experiment.
For this case, only the transition method and the turbulence generator
method can be applied. The domain is not streamwise homogeneous,
ruling out the cyclic method. The boundary layer at the cavity is not
fully developed, making the driver domain method not applicable. The
transition method would model the entire setup, including the spires and
the fetch between the spires and the cavity. The spires need to be resolved
with a fine mesh to capture the turbulence generation process accurately.
This results in a very large cell count, the overhead to achieve the proper
turbulence levels at the cavity is therefore very high. The domain size for
the turbulence generator method is substantially smaller, including only
a short inlet fetch in front of the cavity. The turbulence generator method
however requires a high quality dataset, measured at a location upstream
of the cavity.

Based on this evaluation, the turbulence generator method was selected,
due to the wide range of applicability, high level of control and computa-
tional efficiency. The main drawback of the method is, that the turbulence
at the inlet has a artificial structure and needs to develop into physically
realistic turbulence. This might result in a modification of the specified
mean flow profile, Reynolds stresses and length scales. A comparison to
wind tunnel measurements of the cavity flow will be used evaluate these
deficiencies (Chapter 7.2).

Two turbulent inflow generation methods were considered, the method
of random turbulent spots (Kornev et al. 2008) and the filtered noise
method (Klein et al. 2003). For the turbulent spots method an OpenFOAM
implementation exists (LEMOS 2015). No implementation was available
for the filtered noise method.

Figure 6.2 presents flow field results from a preliminary investigation of
different methods for the cavity case shown in Fig. 6.1. A PIV snapshot is
included to serve as a comparison. The spires were geometrically resolved
and the wake flow simulated. Even tough the turbulent structures are
Figure 6.1: Section of the wind tunnel setup for the turbulent cavity flow experiment. Two Boxes show the outlines of possible computational domains, resolving the spires or using a synthetic turbulence generation respectively.

similar as observed in PIV, the developed mean flow profile was not found to be satisfactory. This was attributed to a too low mesh resolution in the development section behind the spires and the wall. The next panel shows a flow field generated using the turbulent spots code. This code allows for the specification of one integral length scale per velocity. The integral length scales for the streamwise direction were extracted from PIV and used for the simulation. This resulted in very large turbulent structures when compared to PIV. The filtered noise method allows for specification of length scales in all spatial direction for all velocity components. The resulting flow field shows turbulent structures that are more similar in size as observed for the measurements.

In addition, the computational efficiency and the ability to specify different length scales for different heights make the filtered noise BC a good candidate for local scale urban flow simulation (Xie and Castro 2008). Different length scales are required since turbulence near the ground (generated by the rough ground or buildings) is smaller than turbulence in the upper part of the ABL. However, two main difficulties were identified for the application of the filtered noise method. First, the high level of control over the input data requires good knowledge of the flow to be simulated. Secondly, no formal method exists to obtain the parameters of the method (filter coefficients). Consequently, little information exists on
6.1 Evaluation of Methods

Figure 6.2: Comparison of simulated turbulent flow fields between a geometrical inflow generator (spires), the method of turbulent spots and the filtered noise method. Vertical streamwise planes of instantaneous velocity magnitude are shown. The bottom panel shows a snapshot from a PIV measurement. A dashed black box in the upper panels with the same size as the PIV plane is included as a reference.

The effects of the choice of parameters on the generated turbulence. These points will be addressed in this chapter.
6.2 FILTERED NOISE METHOD

The filtered noise method was originally proposed by Klein et al. (2003). They propose to use a digital linear non-recursive filter. Applied to random Gaussian white noise, this generates fields with prescribed autocorrelations.

For each time step of the simulation, a volume of random data $\xi$ is filtered, from which a 2D slice of fluctuating velocity data is extracted. The fluctuating velocity data is superimposed with the desired mean velocity profile and applied to the inlet boundary. For the next timestep, the corresponding slice of random data is then discarded from each volume $\xi$ and a new slice of random data is added. The new random volume is used to filter a new slice of fluctuating velocity data.

6.2.1 Procedure

The procedure is outlined as follows:

1. Generation of normally distributed uncorrelated random numbers $\xi$ with unity variance
2. Imposing a two-point autocorrelation through a filtering operation $v = G * \xi$
3. Scaling to achieve desired variances and covariances $u' = Lv$
4. Superposition with a mean velocity $u = u' + \bar{u}$

The steps 1-3 are visualized in Figure 6.3.

We look at the first three steps in detail. The fourth step, the superposition of the mean flow, is trivial.

Random Vector

The first step is straight forward. For each component of the velocity vector $u_i$, a random field $\xi_i$ is generated, with the properties $\xi_i = 0$ and $\xi_i^2 = 1.0$ and a normal distribution (Gaussian white noise).
6.2 Filtered Noise Method

**Figure 6.3:** Visualization of the outputs for the intermediate steps to generate a fluctuating field with prescribed autocorrelation and variance.

### Filtering

The second step contains the core of the method. With an appropriate choice of the filter kernel $G$, a signal with a prescribed autocorrelation is generated. Since the autocorrelation is related through the Fourier transform to the spectrum, this is equivalent of imposing a desired Power Spectral Density (PSD) (Eq. 3.29).

The filtering operation is defined through the convolution $G * \xi$. The discrete, one dimensional implementation is defined by

$$v_m = \sum_{n=-N}^{N} b_n \xi_{m+n} \quad (6.1)$$

where $b_n$ are the coefficients of the filter $G$ with width $2N+1$ and $\xi$ is a vector of random numbers.

The filter coefficients proposed by Klein et al. (2003) follow the function

$$b_n = e^{-\frac{\pi n^2}{4L^2}} / \tilde{b} \quad (6.2)$$

with a normalizing factor $\tilde{b}$. The resulting autocorrelation function follows a Gaussian

$$\rho(\tau) = e^{-\frac{\pi \tau^2}{4L^2}} \quad (6.3)$$

with $L$ being the integral scale.

The filtered noise procedure can be easily implemented for three dimensions, where volumes of random data $\xi$ are generated instead of...
fields. The subsequent filtering operation is then also performed in 3D. A three dimensional filter can be obtained by a convolution of three one dimensional filters:

\[ b_{ijk} = b_i \ast b_j \ast b_k \]  

(6.4)

**Scaling**

The filtered fields \( v_i \) are still uncorrelated with respect to each other. In order for the velocity to have a desired Reynolds stress, a transformation of the vector \( v_i \) is performed as:

\[ u_i = L_{i,j} v_i \]  

(6.5)

where \( L_{i,j} \) is the lower triangular matrix of the the Cholesky decomposition of the Reynolds Stress tensor \( R \):

\[ R = LL^*. \]  

(6.6)

This procedure takes advantage of the fact that the Reynolds stress tensor is the covariance matrix, and therefore the transformation Eq. 6.5 correlates the variables \( v_i \). The components of \( L \) can be determined recursively by the following algorithm, as described e.g. in Ballard et al. (2009):

\[ L_{j,j} = \sqrt{R_{jj} - \sum_{k=1}^{j-1} L_{j,k}^2} \]  

(6.7)

\[ L_{i,j} = \frac{1}{L_{j,j}} \left( R_{i,j} - \sum_{k=1}^{j-1} L_{i,k} L_{j,k} \right), \quad \text{for } i > j. \]  

(6.8)

For the rank 3 tensor \( R_{i,j} \), the resulting matrix reads:

\[
L_{i,j} = \begin{bmatrix}
\sqrt{R_{11}} & 0 & 0 \\
\frac{R_{12}}{L_{11}} & \sqrt{R_{22} - L_{21}^2} & 0 \\
\frac{R_{13}}{L_{11}} & \frac{(R_{23} - L_{21} L_{31})}{L_{22}} & \sqrt{R_{33} - L_{31}^2 - L_{32}^2}
\end{bmatrix}. \]  

(6.9)

Since this transformation depends only on \( R_{i,j} \), and \( R_{i,j} \) remains constant for statistically stationary flows, it has to be computed only once.
This method was popularized by Lund et al. (1998), who reported the components of \( L \) without prior derivation. Therefore, the method is commonly referred to as the **Lund Transformation**.

### 6.2.2 Simplifications

The small timestep of LES together with the requirement for a long simulation time for statistical averaging results in a large number of timesteps. Unless pre-generated inlet data is used, the inflow generation operation has to be performed for every timestep. Therefore, the method employed and its implementation should be as efficient as possible.

Several authors have proposed simplifications and improvements. In the case of the filtered noise method, the main computational cost comes from the filtering operation.

Veloudis et al. (2007) address the computational cost of the filtering operation, by performing the convolution of the filter and the random field in the frequency domain, using the Fast Fourier Transform (FFT) and inverse FFT. Additionally, they use a reduced number of cells and interpolate onto the simulation. Contrary to the first measure, the second measure reduces the accuracy of the correlation, as an interpolation is basically an additional filtering step.

Kempf et al. (2012) also reduced the computational time of the filtering. Their approach is to perform the filtering operation in three dimensions successively instead of simultaneously. Each direction takes advantage of values computed for the previous direction, reducing the order of computations from \( L^3 \) to \( 3L \). Additionally, they propose a parallel implementation, in which each processor only computes the part of the inlet patch that is assigned to.

Xie and Castro (2008) essentially reduce the method from 3D to 2D. The filtering is still performed in the two non-streamwise directions, however in the streamwise direction, the velocity at time \( t + 1 \) is expressed as a function of the velocity at time \( t \) and a newly generated field.

### 6.3 Present Method

This section describes a modification and extension to the filtered noise inflow boundary condition originally proposed by Klein et al. (2003). A
new procedure to derive the filter kernels for the spatial filtering is presented and the resulting filters are validated. Concepts from time series Autoregressive Moving-Average (ARMA) modelling are used to derive the temporal filtering and create a link between the three dimensional implementation by Klein et al. (2003) and the two dimensional simplification by Xie and Castro (2008).

6.3.1 *Spatial Correlation*

Spatial correlation is achieved using the digital filtering of a random scalar field (Eq. 6.1). For the lateral and vertical direction, this is implemented as:

\[ \psi_{m,n} = \sum_{j=-N_y}^{N_y} \sum_{k=-N_z}^{N_z} b_{j,k} \xi_{m+j,n+k} \]  

(6.10)

with

\[ b_{j,k} = b_j \ast b_k. \]  

(6.11)

being the coefficients of the 2D filter kernel, and \( N_y \) and \( N_z \) relate to the support of the filter. The principal difficulty of the Filtered Noise method lies in obtaining the coefficients \( b_j \) and \( b_k \) of the filter kernel. Klein et al. (2003) proposes to numerically invert

\[ \rho(\tau) = \sum_{j=-N+\tau}^{N} b_j b_{j-\tau} / \sum_{j=-N}^{N} b_j^2 \]  

(6.12)

to find the filter coefficients for a desired correlation function. This is however impractical, since the full correlation function might not be known and a mismatch of grid resolution and lag \( \tau \) requires interpolation. A more convenient form is to define an analytical correlation function with known integral scale and for which an analytical solution for the filter coefficients exist. Klein et al. (2003) states a solution for the filter coefficients (Eq. 6.2) that result in a Gaussian correlation (Eq. 6.3), however without derivation.

The filtering operation (Eq. 6.1) can be recognized as a Finite Impulse Response (FIR) filter. A digital, discrete time FIR filter (Van Etten 2006) takes an input sequence \( x \) and outputs \( y \):

\[ y_n = \sum_{k=0}^{N} h_k x_{n-k} \]  

(6.13)
where $h_k$ is the impulse response of the filter with width $N$. This is the discrete implementation of the convolution (Eq. 3.21). The autocorrelation of the output $y$ is given by the convolution of the autocorrelation of the input $x$ with the autocorrelation of the impulse response $h$ (Girod et al. 2001):

$$\rho_y = \rho_h \ast \rho_x. \tag{6.14}$$

When the input sequence is random noise, the autocorrelation of the input is one for lag zero and zero everywhere else. Therefore, the autocorrelation of the output is equal to the autocorrelation of the filters impulse response $h$. Using the cross-correlation theorem (3.25), the autocorrelation can be written as:

$$\rho_h(\tau) = \mathcal{F}^{-1}\{|\mathcal{F}\{h(\tau)\}|^2\} \tag{6.15}$$

Therefore, an analytical solution for the filter exists if a Fourier transform is known for the impulse response function and an inverse Fourier transform for its absolute square.

Solutions for three functions were found: the single sided exponential

$$h(x) = \begin{cases} e^{-ax}, & x \geq 0 \\ 0, & x < 0 \end{cases} \tag{6.16}$$

the double sided exponential

$$h(x) = e^{-|a|x} \tag{6.17}$$

and the Gaussian:

$$h(x) = e^{-ax^2} \tag{6.18}$$

The solution for the filter coefficients and the corresponding ACF are presented in Table 6.2 with the parameter $a$ expressed by the integral length scale $L$. The solutions are visualized in Figure 6.4.

The Gaussian correlation is equal to the function in Klein et al. (2003). The function for the filter coefficients given by Xie and Castro (2008) for an exponential Autocorrelation Function (ACF) correspond to the double exponential function (Tab. 6.2). The filter coefficients that actually result in a exponential ACF are obtained by the (single sided) exponential function.

The parameter $\tilde{b}$ in Table 6.2 is used to normalize the standard deviation and is defined by Klein et al. (2003) as:

$$\tilde{b} = \sqrt{\frac{\sum_{i=0}^{N} b_i^2}{N}} \tag{6.19}$$
Figure 6.4: Filter coefficients (left) for three target autocorrelation functions (right). The markers show the analytical solution, the lines the results from the simulation. The integral scale is $20\tau$.

Table 6.2: Spatial Filters. The parameter $L$ is the integral length scale in grid units $\tau$.

<table>
<thead>
<tr>
<th>Filter Name</th>
<th>Filter Coefficients $b_j$</th>
<th>ACF $\rho(\tau)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exponential</td>
<td>$e^{-\frac{\bar{L}}{\bar{b}}}$, $j &gt; 0$</td>
<td>$e^{-</td>
</tr>
<tr>
<td>Double Exponential</td>
<td>$e^{-2</td>
<td>\frac{</td>
</tr>
<tr>
<td>Gaussian</td>
<td>$e^{-\frac{\pi^2}{4\bar{L}^2}}/\bar{b}$</td>
<td>$e^{-\frac{\pi^2}{4\bar{L}^2}}$</td>
</tr>
</tbody>
</table>

A one-dimensional numerical calculation was performed to validate the correct shapes of the two-point correlation functions. In the one-dimensional case, the two-point correlation corresponds to the (spatial) ACF. Random numbers are generated and filtered using the three defined filter kernels (no flow equations are solved). A length scale of $L = 20\tau$ was chosen. The computed ACF is presented in Figure 6.4 together with the values of the filter coefficients. For all three filter kernels the desired target ACF is achieved.
6.3.2 Temporal Correlation

The streamwise correlation is performed in the time domain by using an Autoregressive (AR) model of order one (Eq. 3.42). The AR(1) ACF is exponential (Eq. 3.44) and the AR(1) parameter can be determined (Eq. 3.46) in terms of the integral time scale as

\[ \varphi = e^{-\frac{\Delta t}{T}}. \] (6.20)

where \( \Delta t \) is the simulation timestep.

In order to use the Cholesky transformation (Eq. 6.5) to obtain the desired Reynolds stress tensor, the output of the AR model needs to have unity variance. The model is therefore modified by scaling the variance with \( c \):

\[ \sigma^2_{X^*} = \frac{\sigma^2_X}{c} = 1 \] (6.21)

where \( \sigma^2_X \) is the variance of the AR(1) model (Eq. 3.43). Substituting \( \sigma^2_X \), this is equal to an appropriate scaling of \( \sigma^2_\varepsilon \) with \( c \):

\[ c \sigma^2_\varepsilon = 1 - \varphi^2. \] (6.22)

If the variance of the innovation is one (\( \sigma^2_\varepsilon = 1 \)), the scaling parameter \( c \) can be written as

\[ c = 1 - \varphi^2 \] (6.23)

Using the definition of the variance (Eq. 3.5), it can be shown that scaling of the variance of the innovation (lhs of Eq. 6.22) is equivalent to scaling the random innovation term \( \varepsilon_t \) by \( \sqrt{c} \):

\[ c \frac{1}{N} \sum_{k=0}^{N} (\varepsilon_k)^2 = \frac{1}{N} \sum_{k=0}^{N} (\varepsilon_k \sqrt{c})^2 \] (6.24)

and finally resulting in the modified AR(1) model where the innovation term \( \varepsilon_t \) is scaled to result in \( \sigma^2_{X^*} = 1 \):

\[ X^*_t = \varphi X^*_{t-i} + \varepsilon_t \sqrt{1 - \varphi^2} \] (6.25)

Combining Eq. 6.20 and 6.25, the final implementation for the temporal correlation of a new slice of an unscaled field of fluctuating velocity reads

\[ u_t = u_{t-1} e^{-\frac{\Delta t}{\overline{t}}} + v_t \sqrt{1 - e^{-2\Delta t/\overline{t}}}, \] (6.26)

with the innovation term \( \varepsilon_t \) replaced by the random filtered velocity field \( v_t \) and \( X_t \) with \( u_t \).
Figure 6.5: Steps of the filtered noise inflow generation method. An exemplary filter kernel is displayed as a red rectangle with a red arrow pointing to the location of the target cell.

6.3.3 Implementation

Figure 6.5 schematically visualizes the steps of the inflow generator. Two different types of grids are used, a virtual grid and a patch grid. The virtual grid has a uniform grid spacing whereas the patch grid corresponds to the inlet patch of the computational domain. Two virtual grids are used, one to hold the random field and one to hold the filtered field. The extent of the grid for the random field is larger than that for the filtered field, by half the filter kernel size in each direction. As the filtered field is interpolated from the virtual grid onto the patch grid, the virtual grid spacing should correspond to the smallest cell size of the patch grid.

The FilteredNoise inflow generator was implemented in C++ for the Computational Fluid Dynamics (CFD) code OpenFOAM (Sec. 5.4) as a BC library (Immer and Vonlanthen 2015a). Having the code as a library allows the boundary condition to be used in any application written
within the *OpenFOAM* framework. Input data is provided in the form of unstructured grid data and is interpolated onto the respective grids. This allows to provide arbitrarily, spatially varying fields for the mean velocity, Reynolds stress tensor, integral time scale and integral length scales. A minimum of four points, at each corner of the bounding box of the inlet patch, are needed to impose a constant value. For typical boundary layer simulations, vertical profiles can be imposed by providing two vertical lines of data at each side of the inlet patch. The imposed data can be arbitrarily spaced, facilitating the use of third party data (either from measurements of simulation).

The main features of the *FilteredNoise BC* code are summarized as follows:

- Implemented as a velocity boundary condition library
- Parallel implementation
- Restart capability
- Three spatial correlation functions: exponential, double exponential and Gaussian
- Exponential correlation in time direction
- Turbulent scalar field generation
- Mass flow correction

Parallelization is handled by *OpenFOAM* by decomposing the domain into a number of subdomains corresponding to the number of processors. The code is then executed in parallel and the communication between the domains is handled through a Message Passing Interface (*MPI*). The consequence of this is that even if a domain is decomposed into a large number of subdomains, only a small number of subdomains may contain the inlet boundary patch. This reduces the potential gain in speed-up of the boundary code. Therefore, the *FilteredNoise BC* handles the parallelization of filtering operation separately: every processor is assigned an equal number of virtual grid points. Since every point has one filter kernel attached, the filter coefficients are computed on the respective processors, taking advantage of the full number of processors. Once the inflow data is mapped to the inlet patch, the default *OpenFOAM* parallelization through the domain decomposition is used.
The restart capability is important, if a run cannot be completed in one execution, or if a simulation is composed of multiple phases (e.g. initialization and sampling). In order to restart the simulation from a previous timestep $t-1$, Equation 6.26 shows that only the time correlated, filtered patch field $u_{t-1}$ (prior to the scaling step) is required. During a run, this field is kept in memory and is written to disk at the last timestep.

A correction of the final fluctuating velocity field is performed in order to keep the mass flow imposed by the mean velocity field constant. This eliminates large global pressure fluctuations.

Turbulent scalar field generation is presented separately in Section 6.5.

6.3.4 Discussion

The presented method uses an AR(1) model for the temporal correlation. A crucial step to improve the usability of the implementation is to express the AR(1) parameter $\phi$ through a closed-form solution. The AR(1) parameter $\phi$ is expressed as a function of the integral time scale, instead of through the autocorrelation $\rho$ at lag $\tau = 1$. This removes $\phi$’s dependency on the simulation timestep $\Delta t$, allowing to provide input data (integral times) to the FilteredNoise inflow generator independent of the timestep of the simulation. The final function (Eq. 6.26) is equivalent to the two-dimensional simplification proposed by Xie and Castro (2008), however was independently derived from an AR model.

The main drawback of this simplification is the restriction to an exponential ACF. Turbulence does not show an exponential decay of $\rho$ at small lags, but rather a smooth curvature called the Taylor Microscale (Kundu and Cohen 2001). As the present method was derived from an AR model, it is not restricted to an exponential ACF (AR(1) model), any higher order AR(p) models can be used. Increasing the order of the model allows for control over multiple points of the ACF, and therefore over the curvature around $\tau = 0$. This enables a more realistic model for turbulence.

The second order model AR(2) allows $\rho$ to be specified for lags $\tau = 1$ and $\tau = 2$ (Fig. 6.6). The equation for the AR(2) time series model reads:

$$X_t = \phi_1 X_{t-1} + \phi_2 X_{t-2} + \epsilon_t,$$

(6.27)

with variance

$$\sigma^2_X = \frac{(1 - \varphi_2)\sigma^2_\epsilon}{(1 + \varphi_2)((1 - \varphi_2)^2 - \varphi_1^2)}.$$

(6.28)
As for the \( AR(1) \) model, a closed-form solution of the \( ACF \) can be derived for the \( AR(2) \) model:

\[
\rho(\tau) = \frac{(1 - \alpha_2^2)\alpha^{\tau+1} - (1 - \alpha_1^2)\alpha_2^{\tau+1}}{(\alpha_1 - \alpha_2)(1 + \alpha_1\alpha_2)},
\]  

(6.29)

with

\[
\alpha_1 = \frac{-2\varphi_2}{\varphi_1 + \sqrt{\varphi_1^2 + 4\varphi_2}},
\]

(6.30)

\[
\alpha_2 = \frac{-2\varphi_2}{\varphi_1 - \sqrt{\varphi_1^2 + 4\varphi_2}}
\]

(6.31)

Equations for the two \( AR(2) \) parameters \( \varphi_1 \) and \( \varphi_2 \) can easily be derived from these equations. An experimental implementation of an \( AR(2) \) based \textit{FilteredNoise} code is available (Immer and Vonlanthen 2015a).

For practical applications, the \( AR(2) \) model suffers from numerical accuracy problems. If the integral timescale is much larger than the timestep (\( T \gg \Delta t \)), \( \varphi_1 \) and \( \varphi_2 \) can be nearly equal. Due to the limited numerical accuracy, the resulting \( ACF \) can vary greatly from the desired shape and therefore the integral time scale will not match the required value. A way to solve this problem is to use use a higher order \( AR(p) \) model, to capture the \( ACF \) with sufficient points. Instead of a closed-form
solution for the AR parameters, the p parameters $\varphi$ can be recursively computed using the Yule-Walker equations (Box et al. 2008). When the entire ACF is captured (up to a limit, e.g. four times the integral scale), the AR(p) model is equivalent to the three dimensional filtering proposed by Klein et al. (2003). The choice of the order p allows to adjust the balance between the control over the shape of the ACF and the computational efficiency.

The spatial filtering was presented in a digital filtering form and closed-from solutions for the filter coefficients were presented for three correlation functions. The non-recursive digital filter (Eq. 6.1) is equivalent to an MA(q) model, where the filter coefficients $b_i$ correspond to the Moving-Average (MA) parameters $\theta_i$:

$$X_t = \sum_{i=1}^{q} \theta_i \varepsilon_{t-i}$$

An AR(p) model can be converted into a MA(\infty) model (Box et al. 2008). The MA(q) parameters $\theta_i$ can therefore easily be found using the methods for the AR(p) model and subsequently truncating the MA(\infty) model. The order q corresponds to the desired filter kernel width.

The presented method and implementation of the filtered noise inflow generator is intended to be used with the LES approach. However, there is no restriction in the method that would prevent the use for a Direct Numerical Simulation (DNS), hybrid LES or even an unsteady Reynolds-Averaged Navier-Stokes (RANS) computation.

### 6.4 Application to Boundary Layer

A set of simulations on a turbulent flat plate boundary layer was performed using the LES model described in Chapter 5. The ability of the FilteredNoise BC to provide the specified turbulent statistics is tested and the effect of the BC’s input parameters on the evolution of the flow downstream of the inlet is investigated.

The required input data was obtained from the time resolved stereo PIV measurement of the cavity experiment (Chapter 4). The available PIV data provides vertical profiles of autocorrelation coefficients and profiles of vertical two-point correlations, allowing to extract six of the nine required integral length scales. The missing scales were reconstructed.
6.4 Application to Boundary Layer

6.4.1 Computational Setup

The computational domain (Figure 6.7) for the flat plate is a box with extent \( x \times y \times z = 6H \times 2H \times 1.6H \) where \( H \) is the boundary layer height \((H = 0.1 \text{ m})\). A base mesh with streamwise aligned hexahedrals cells with size \(0.006 \text{ m} \times 0.004 \text{ m} \times 0.004 \text{ m}\) is created (Level 0). Subsequent refinement is performed by cutting each side in half, resulting in 8 smaller cells with equal size. The mesh refinement regions are shown in Figure 6.7, with the base cell size at Level 0 and each successive refinement denoted \( L_1 \) to \( L_3 \). Level 1 was chosen to include the entire turbulent boundary layer. The next finer level \((L_2)\) was set at \( y = 0.05 \text{ m}\). The additional level \( 3 \) is a wall refinement layer. This final mesh counts \( 4'168'000 \) cells.

The boundary conditions are set according to Sec. 5.3.6. For the inlet boundary, the FilteredNoise code as described in Section 6.3 is used.

For the subgrid Turbulent Kinetic Energy (TKE), a small value of \( \kappa_{sgs} = 0.0005 \text{ m}^2/\text{s}^2 \) is imposed at the inlet. In addition, a passive scalar was released a laterally extending line near the ground and inlet \((x = 0.0098 \text{ m} \text{ and } y = 0.0027 \text{ m})\) to serve as a tracer for the initialization run. For all fields, the lateral boundaries are periodic (laterally homogeneous flow).

Simulations were performed at the Reynolds number \( Re_\delta = 20'260 \), based on the free stream velocity \( u_\infty = 3.14 \text{ m/s} \), the boundary layer height \( \delta = 0.1 \text{ m} \) and the kinematic viscosity \( \nu = 1.55 \times 10^{-5} \text{ m}^2/\text{s} \). The timestep was set to \( \Delta t = 1.25 \times 10^{-4} \text{ s} \), resulting in a mean Courant number of \( Co_{\text{mean}} \approx 0.1 \) and a maximum of \( Co_{\text{max}} \approx 0.3 \).

Each simulation is divided into an initialization phase and a sampling phase, according to Section 5.3.7. The scalar tracer (released at the inlet near the wall) is first detected at the outlet after 0.318 seconds (extracted from the case \( FN_{\text{exp}} \)). This can be interpreted as the time it takes for the inflow data to flush the entire domain and is therefore the absolute minimum initialization time. Convergence of the scalar field was observed after 1.5 seconds, and with an additional margin the initialization was run for a total of 2 seconds. With a free stream velocity \( u_\infty = 3.15 \text{ m/s} \) and the domain length of 0.6 \text{ m}, this results in about 10.5 equivalent flow through times. This is a significantly lower initialization time in comparison to the \( 10^4 \) flow through times needed for the periodic channel flow of Kravchenko et al. (1996).

Statistics were collected over a duration of 60 seconds. With an average integral time scale of \( T = 0.01 \text{ s} \) in the boundary layer, this results in 3000
independent samples. This makes the statistical uncertainty comparable to the PIV measurements.

6.4.2 Inlet Data

The required inlet data are extracted from time resolved and time averaged Stereo-PIV measurements on a vertical streamwise plane for the case 10m (Chapter 4). The case 10m was chosen because of the large thickness of the boundary layer (not just influenced by the wall but also by the spire generated turbulence, while still fully visible in the PIV field of view) and the higher degree of turbulence than the narrow spire case 10n while showing only a small lateral inhomogeneity (compared to the case 10w in Figure 4.11).

The required first order statistics, profiles of mean velocity $\bar{u}(y)$ and Reynolds stress tensor $R_{ij}(y)$, were generated directly from the time averaged PIV measurements. The Stereo-PIV measurement provides all six components of the Reynolds stress tensor. A spline fitting was used to smooth the PIV data and generate missing near wall data points.

Simplified profiles for the integral time- and length scales were estimated from the measured autocorrelations and two-point correlations,
respectively. From the time resolved measurement, autocorrelations are extracted and by using the $1/e$ method described in Section 4.3.4, the integral time scales are computed. The time scales are converted to length scales using the local mean velocity $u_1$ (Eq. 3.40). For the length scales in vertical direction $L_{i,y}$, the two-point correlations are computed using the PIV snapshots from the time averaged measurement (uncorrelated snapshots). The length scale was computed in positive vertical direction (3.36) using the $1/e$ method. For the length scales in z-direction $L_{i,z}$, PIV data is not available. Therefore, the lateral length scales were estimated from two point correlations taken from the DNS of Moser et al. (1999). From these nine profiles of length scales, simplified three zone profiles were created. The final input dataset $10m\_simple$ is shown in Figure 6.8.

A second input dataset was created by extracting the nine length scales directly from a LES computation, after a development length of 0.5 m (Fig. 6.7). The autocorrelations and vertical two point correlations were obtained from the center line point probes and the lateral length scales from the xy-plane point probes. The new input dataset $10m\_adapted$ is shown in Figure 6.9. The flow statistics and the time scales (length scales in streamwise direction) are identical to the $10m\_simple$ dataset.

6.4.3 Cases

Multiple simulations were run with the FilteredNoise inflow boundary condition. Three cases with different integral time scales were simulated to show the influence of the streamwise correlation on the flow development: one baseline case ($FN\_exp$), one case with a five larger time scale ($FN\_T_5$) and one with a factor of five smaller time scale ($FN\_TFifth$). These cases represent cases where the timescale is either significantly overestimated or underestimated when creating an input dataset. For these cases, the spatial correlation function is exponential (filter kernel size $N = 2L$). Simulations with three different spatial correlation functions were performed (filter kernel size $N = 4L$): exponential ($FN\_exp$), double exponential ($FN\_dblExp$) and Gaussian ($FN\_gauss$), as specified in Table 6.2. An additional case with Gaussian correlations ($FN\_gauss\_adp$) was simulated with adapted spatial length scales (Fig. 6.9).
Figure 6.8: Boundary layer input data for the FilteredNoise BC, from the PIV case 10m, with mean velocity $\bar{u}_x$, Reynolds stresses $R_{ij}$ and integral length scales $L_{i,j}$.

Table 6.3: Turbulent boundary layer LES cases

<table>
<thead>
<tr>
<th>Case</th>
<th>Spatial Correlation</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>FN_T/5</td>
<td>Exp.</td>
<td>$T_{i,x}/5$</td>
</tr>
<tr>
<td>FN_5T</td>
<td>Exp.</td>
<td>$5T_{i,x}$</td>
</tr>
<tr>
<td>FN_exp</td>
<td>Exp.</td>
<td></td>
</tr>
<tr>
<td>FN_dblExp</td>
<td>Dbl. Exp.</td>
<td></td>
</tr>
<tr>
<td>FN_gauss</td>
<td>Gaussian</td>
<td></td>
</tr>
<tr>
<td>FN_gauss_adp</td>
<td>Gaussian</td>
<td>Adapted spatial scales</td>
</tr>
</tbody>
</table>

All cases use the AR(1) model for time correlation, and therefore have exponential ACFs, independent of the spatial correlation shape. The complete list of simulations is presented in Table 6.3.
6.4 Application to Boundary Layer

Figure 6.9: Boundary layer input data for the FilteredNoise BC, from the PIV case 10m, with mean velocity $\bar{u}_x$, Reynolds stresses $R_{ij}$ and integral length scales $L_{i,j}$. The vertical ($L_{i,y}$) and lateral ($L_{i,z}$) scales are adapted from the LES case FN_gauss_adp.

6.4.4 Results

The instantaneous velocity field developing in the domain is presented in Figure 6.10 for the case FN_exp. The horizontal velocity $u_x$ shows clearly the (instantaneous) interface between the turbulent boundary layer flow and the laminar free stream. Between $x = 0 \ m$ and $x = 0.1 \ m$, nonphysical structures in the form of horizontal streaks are visible, presumably due to the fine grained structures resulting from the exponential filtering. These disappear further downstream, and no visible change in the turbulent structures is apparent. The vertical velocity field $u_y$ shows smaller structures towards the wall and larger structures towards the boundary layer top. The structures in the upper boundary layer seem finer at the inlet and get progressively coarser further downstream. Occasional very large, weak areas of vertical velocity extend into the free stream up to the top ($y = 0.2 \ m$), visible e.g. between $x = 0.1 \ m$ and $x = 0.2 \ m$ (upwards motion) or between $x = 0.5 \ m$ and $x = 0.6 \ m$ (downwards motion).
Figure 6.10: Instantaneous velocity for the case $FN_{\text{exp}}$ on a vertical streamwise centerline plane ($z = 0$).

The lateral velocity field $u_z$ initially shows circular smaller structures ($x = 0.0 - 0.1 \, m$). Further downstream, distinct elongated, upwards tilted structures appear at the wall. These can be associated with coherent structures forming near the wall. Figure 6.11 visualizes these coherent structures by using the instantaneous Q-criterion, computed using the full three dimensional velocity field, for the same snapshot as Figure 6.10. The small, circular structures at the inlet are clearly visible. They develop downstream into elongated, upwards tilted hairpin-vortex-like structures, as described by Adrian (2007).

The non-dimensional distance of the first cell at the wall is shown in Figure 6.12 for the case $FN_{\text{exp}}$. $y^+$ lies between 1.6 and 1.8 for most of
6.4 Application to Boundary Layer

Figure 6.11: Iso-surfaces of instantaneous positive $Q$, colored by the distance from the wall for the case $FN_{exp}$.

Figure 6.12: Non-dimensional distance $y^+$ of the first cell, for the case $FN_{exp}$.

the boundary layer. Since the first cell point is therefore in the viscous sublayer, the wall gradient can be extracted from the mean flow field $\overline{u}_1$ by using the velocity at the first cell. The velocity gradient is used to compute the wall shear stress $\tau_w$:

$$\tau_w = \nu \rho \left( \frac{\partial \overline{u}_1}{\partial y} \right)_{y=0}$$

(6.33)

where $\nu$ is the kinematic viscosity ($1.55 \times 10^{-5} \text{m}^2/\text{s}$) and $\rho$ the density. The wall friction coefficient is then calculated using:

$$C_f = \frac{\tau_w}{0.5 \rho u_\infty^2}$$

(6.34)
where $u_\infty$ is the free stream velocity.

A changing wall friction would mean that the boundary layer is adapting to the wall conditions, whereas a constant (or slowly decreasing) friction coefficients means that the mean flow profile matches the wall conditions.

The wall friction coefficient $C_f$ is presented in Figure 6.13. All cases show a rapid drop in wall friction immediately after the inlet, followed by a gradual increase. The effect of the outlet boundary condition is visible for all cases as an increase in $C_f$, immediately before the exit. The location of minimum $C_f$ lies around $x \approx 0.2$ for all cases, except for the case $FN_{T/5}$. The cases $FN_{exp}$ and $FN_{gauss\_adp}$ show the slowest increase in $C_f$, with an indication of stabilization around $x = 0.5$. The case $FN_{gauss\_adp}$ stabilizes at a higher level. The case $FN_{dblExp}$ and $FN_{gauss}$ show a similar $C_f$, with the minimum slightly later and lower for the case $FN_{gauss}$.

The fast initial drop in skin friction can be interpreted as an adaptation of the imposed mean velocity to the wall conditions. All cases initially show the same rate of decrease in $C_f$. PIV data was not available at the wall and the wall gradient had to be reconstructed. This makes it likely that the imposed mean velocity gradient at the wall does not match the conditions (smooth wall) of the simulation. However, the location of the minimum $C_f$ can be used as an indication of the length it takes the artificial turbulence to change. Based on that observation, all
cases except the case $FN_T/5$ would have a similar development length. However, not only the location of minimum $Cf$, but also the evolution of the skin friction after the minimum should be considered. Generally, a gradually increasing skin friction is an indication that the turbulence in the boundary layer is still adapting to the wall. A naturally developing fully turbulent flat plate boundary layer would show a decrease of skin friction with increasing distance (Schlichting 1979) or if fully developed (at large distance) a constant skin friction. With this criteria, the case $FN_gauss_{\text{adp}}$ showing a higher level of $Cf$ and only a slow rise performs the best. The case without adapted scales ($FN_gauss$) and the case $FN_{\text{dblExp}}$ reach the same final value, while the case $FN_{\text{exp}}$ seems to stabilize at a lower level. The different wall frictions show, that that the choice of length scale at the wall has an important influence to reach a fast stabilization of $Cf$. Furthermore, the choice of correlation function (Gaussian or double exponential) can give the same final $Cf$ value, however needing a longer distance. The large change of the timescale $T$ shows that longer timescales seem to give better results than too short timescales.

To further investigate the turbulence development at the wall, the instantaneous horizontal velocity $u_x$ for a horizontal plane at $y = 0.00175$ m (corresponds to $y^+ \approx 10$, just above the viscous sublayer with $y^+ > 5$) is presented in Figure 6.14. All cases show long streamwise streaks of high velocity, laterally spaced apart by $\approx 0.02$ m. These streaks are typical for a turbulent boundary layer and have been visualized by Kline et al. (1967). The streak patterns observed towards the end of the domain look similar for all cases, except for the case $FN_T/5$. This case shows a significantly lower velocity, consistent with the lower wall friction (Fig. 6.13). Next, we compare the lateral spacing of the artificial structures generated at the inlet with the lateral spacing of streaks further downstream for different spatial correlation functions. The cases $FN_{\text{exp}}, FN_{\text{dblExp}}$ and $FN_{\text{gauss}}$ (Fig. 6.14a, d and e) show progressively larger lateral spacing between the artificially generated structures at the inlet. The spacing of the artificial structures at the inlet observed for the case $FN_{\text{exp}}$ is smaller than the streak spacing towards the end of the domain. For the case $FN_{\text{dblExp}}$, the spacing of the artificial structures at the inlet is larger than the streak spacing further downstream. The case $FN_{\text{gauss}}$ also shows a larger spacing at the inlet and even larger than the case $FN_{\text{dblExp}}$. The streak pattern at the beginning of the domain ($x = 0.0$ m $- 0.05$ m), visualized through the horizontal velocity, therefore depends on the correlation shape. The case
Figure 6.14: Instantaneous horizontal velocity $u_x$ for a horizontal plane at $y = 0.00175 \text{ m} \ (y^+ \approx 10)$.

$FN_{\text{gauss adp}}$ (Fig. 6.14f) shows a streak spacing at the inlet that is similar to the streak spacing further downstream. This is achieved through the adapted, smaller lateral length scales compared to the case $FN_{\text{gauss}}$.

The velocity fields on the inlet patch, as generated by the filtered noise method, are shown in Figure 6.15 for exponential, double exponential
6.4 Application to Boundary Layer

and Gaussian spatial correlations. All velocity fields clearly show the effect of the respective filtering: the exponential filter shows small, fine grained structures (Fig. 6.15a) and the Gaussian filter shows large, smooth structures (Fig. 6.15c). The double exponential case lies in between the two other cases, featuring smoother structures that the exponential case, but still showing some of the finer detail (Fig. 6.15b). Due to the constant length scale profiles (Fig. 6.8), no vertical variation of size appears. For the case with the adapted length scales (FN_gauss_adp), the effect of the linearly growing length scales (Fig. 6.9) on the velocity field is clearly visible in Figure 6.15d. Small structures near the wall grow towards the boundary layer top. For all cases, the scaling of the Reynolds stresses is clearly visible in the magnitude of the velocity components, particularly for the vertical and lateral velocity (u_y and u_z). The reduction in magnitude towards the boundary layer height (y = 0.1 m) is according to the input profiles of u'_1u'_1, u'_2u'_2 and u'_3u'_3 (Fig. 6.8).

The size of the generated turbulent structures near the wall is consistent to the different patterns of artificial turbulent structures developing after the inlet in Figure 6.14. The size of the generated structures near the wall for the case FN_exp is comparable with the case FN_gauss_adp, this is also visible in the the similarities of the streak patterns at the inlet observed in Fig. 6.14a and f.

To verify that the (lateral and vertical) scales of the generated turbulent structures, as visible in Figure 6.15c and 6.15d, correspond to the specified length scale profiles (Fig. 6.8 and Fig. 6.9), the length scales were computed for the cases FN_gauss and FN_gauss_adp. For the sample locations in the first cell (at the inlet), the spatial two-point correlations were extracted from the sampled time series data. The integral length scales were subsequently computed from the spatial two-point correlations for each velocity component, by fitting a Gaussian function and integrating analytically. The resulting scales for the two cases (FN_gauss and FN_gauss_adp) are presented in Figure 6.16. The vertical length scales L_i,y can only be computed approximately, due to the inhomogeneity of the vertical length scale profiles. The sharp step in 6.16a at the wall can therefore not be captured and due to the free stream with virtually no turbulence and small vertical length scales, the values of L_i,y above y ≈ 0.08 m are not reliable. The lateral scales L_i,z can be computed more accurately, since the flow is laterally homogeneous. Therefore, the step (6.16b) is better captured. Overall, for both cases the computed scales match the imposed scales well.
Figure 6.15: Instantaneous velocity for the inlet patch ($x = 0.0 \ m$) for different spatial correlation functions.
Figure 6.16: Imposed (lines) and computed (markers) vertical and lateral integral length scales at the inlet for (a) the case \textit{FN\_gauss} and (b) the case \textit{FN\_gauss\_adp}. (within the statistical accuracy). The results of the adapted profiles (Fig. 6.16b) show that the implemented \textit{FilteredNoise BC} can handle spatially varying filter kernels without problems.

The spatial evolution of \textit{TKE} is presented in Figure 6.17. All cases show an increase of \textit{TKE} at the wall with increasing downstream distance \( x \), due to the development of the wall turbulence. A horizontal division is visible at \( y = 0.05 \) \( m \), with lower \textit{TKE} above and higher \textit{TKE} below. This line marks the mesh refinement zone border between level \( L1 \) and \( L2 \) (Fig. 6.7). A coarser grid resolution therefore has a visible, but only weak effect on the \textit{TKE} development.

A clear difference in \textit{TKE} is visible between the cases \textit{FN\_exp}, \textit{FN\_T/5} and \textit{FN\_5T} (Fig. 6.17a-c). A reduction of the integral time scale by a factor of five compared to the baseline case (\textit{FN\_exp}) results in a significant drop of \textit{TKE}. In contrast, a five times larger integral time scale (Fig. 6.17c)
A horizontal line is extracted at $y = 0.045$ and the evolution of normalized $\text{TKE}$ is presented in Figure 6.18. $\text{TKE}$ is normalized by the value $TKE_{\text{input}}$, extracted from the input dataset (Fig. 6.8) at the same height. The plotted value $TKE / TKE_{\text{input}}$ therefore represents a loss factor. The circular markers show the loss of $\text{TKE}$ in the first cell. Most cases show a loss of $\approx 12\%$ in the first cell, the case $\text{FN}\_\text{exp}$ a loss of $19\%$ and the case $\text{FN}\_\text{T/5}$ a loss of up to $30\%$. A further drop in $\text{TKE}$ is observed in the region $x < 0.2 \, \text{m}$. This region has been previously identified as the zone where the artificial turbulence develops into physical turbulence. The largest decrease of $\text{TKE}$ is observed for the case $\text{FN}\_\text{T/5}$, dropping to below $30\%$ of the imposed $\text{TKE}$. The level of $\text{TKE}$ in the domain ($x > 0.2 \, \text{m}$) is consistent with the observations made for Figure 6.17. However, a more clear difference between the cases $\text{FN}\_\text{dblExp}$, $\text{FN}\_\text{gauss}$ and $\text{FN}\_\text{gauss}\_\text{adapt}$ is visible. The $\text{TKE}$ for the case $\text{FN}\_\text{dblExp}$ decreases to a lower level than the cases with Gaussian correlations. The case $\text{FN}\_\text{gauss}\_\text{adapt}$ shows a stabilization of $\text{TKE}$ already after $x = 0.8$.

The downstream evolution of $\text{TKE}$ and the non-zero Reynolds stress components is presented in Figure 6.19 for the case $\text{FN}\_\text{gauss}\_\text{adapt}$. Regularly spaced vertical profiles were extracted. The vertical profile at $x = 0 \, \text{m}$ is extracted from the statistics on the inlet patch and matches the statistics required by the input data (black dashed line). This verifies that the implemented Cholesky transformation works as intended.

The $\text{TKE}$ of the input data (from PIV) shows two peaks, one near the wall and one further up in the middle of the boundary layer (at $y = 0.05 \, \text{m}$). A pronounced difference between the imposed $\text{TKE}$ at the inlet ($x = 0.0 \, \text{m}$) and the $\text{TKE}$ at $x = 0.1 \, \text{m}$ is visible, the magnitude of the upper peak is reduced. The downstream evolution shows that this peak is not further reduced, $\text{TKE}$ remains constant (between $y = 0.04 \, \text{m}$ and $y = 0.06 \, \text{m}$) or grows very slowly (above $y = 0.06 \, \text{m}$). The lower peak near the wall
Figure 6.17: TKE on a vertical streamwise centerline plane ($z = 0$).
Figure 6.18: Evolution of normalized TKE with downstream distance \( x \), at \( y = 0.045 \) (in the refined mesh region). TKE is normalized by the value from the input data (\( TKE_{\text{input}} \)). The circular markers show the value at the first cell.

However does not show a decrease of TKE: TKE continuously increases further downstream, due to the development of wall turbulence. Towards the end of the domain (\( x = 0.5 \) m), the minimum between the two peaks disappears and the upper peak of TKE is not clearly visible anymore.

TKE is the sum of the three diagonal components of the Reynolds stress (\( u'_i u'_i \)). Next we compare the \( u'_1 u'_1 u'_2 u'_2 \) and \( u'_3 u'_3 \) curves to the observations made for the TKE curve. All three curves show a growth of fluctuations near the wall, due to the developing wall turbulence. A decrease of the upper peak is only observed for the lateral and vertical components (\( u'_2 u'_2 \) and \( u'_3 u'_3 \)). The upper peak the streamwise fluctuations \( u'_1 u'_1 \) decreases only slightly.

The shear stress (\( -u'_1 u'_2 \)) also shows a decrease after the inlet (\( x = 0.1 \)). The upper peak however is largely recovered after at \( x = 0.2 \) m and remains stable. In the lower half, the shear stress matches the imposed value at \( x = 0.2 \) m but then further grows. Near the wall, a profile develops and no further change is observed below \( y = 0.01 \) after \( x = 0.4 \) m, indicating that the flow has adjusted near the wall.

Figure 6.20 shows the comparison between the theoretical spectrum of the AR(1) model (exponential autocorrelation) and the computed spectrum.
Figure 6.19: Profiles of $TKE$ and Reynolds stress components along vertical lines for the case $FN_{gauss\_adp}$. The black dashed lines show the imposed statistics for the generated velocity fields.
in the first cell after the inlet, at $y = 0.038 \text{ m}$. The effect of larger (case $FN_{5T}$) or smaller (case $FN_{T/5}$) integral time scales is clearly visible. The larger the time scale, the more TKE is at lower wavenumbers (lower frequency). The effect of the LES filtering is visible by a decrease of spectral power above a certain wavenumber. This location has been identified as six times the cell size $\Delta x$ and is consistent for all three velocity components (marked by vertical lines). This wave number is likely to correspond to the effective filter width $\Delta_{eff}$ (Fig. 5.1). When integrating the theoretical PSD for $u_x$ from zero to $\Delta_{eff}$, the following maximum percentages of TKE can be captured by the grid: 81.5%, 96.2% and 99.2% (with increasing time scale, for the cases $FN_{T/5}$, $FN_{exp}$ and $FN_{5T}$). The observed loss of TKE in the first cell can therefore not be attributed only to the limited ability of the LES grid to resolved TKE, as the grid of the case $FN_{5T}$ should be able to capture almost all of the imposed TKE.

The time scale and the (exponential) ACF is the same for the cases $FN_{exp}$, $FN_{dblExp}$, $FN_{gauss}$ and $FN_{gauss\_adp}$. Therefore the same spectra as for the case $FN_{exp}$ are observed. The spatial spectra however differ according to the filter function used. The spatial (lateral) spectra for the vertical velocity $u_y$ are presented in Figure 6.21 for the cases $FN_{exp}$,
Figure 6.21: Downstream evolution of lateral velocity spectra of $u_y$, sampled at $y = 0.038$. The dashed lines for $x = 0.0 \, m$ are the theoretical spectra. The dashed black line marks 6 times the cell size ($6\Delta z$).

$FN_{dblExp}$, $FN_{gauss}$ and $FN_{gauss\_adp}$. As expected from the validation (Fig. 6.4), the spectra at $x = 0 \, m$ match the theoretical spectra for all three correlation functions. The effect of the LES filter can be observed at high wavenumbers, as was in Figure 6.20. The evolution of the spectra with increasing downstream distance is visualized by the plots at $x = 0.1 \, m$, $x = 0.2 \, m$ and $x = 0.3 \, m$. The spectra converge at $x = 0.3 \, m$ to the same curve, independent of the initial correlation shape. No further significant changes in the normalized spectra can be observed for distances higher than $x = 0.3 \, m$. The rate of convergence for the cases is consistent with the previous observations for the flow field and the TKE. The case with the adapted scales and Gaussian correlation ($FN_{gauss\_adp}$) converges the fastest, almost no change is visible after $x = 0.1 \, m$. In contrast, the case with the same Gaussian correlation but a larger length scale ($FN_{gauss}$) converges the slowest. The case with the exponential correlation ($FN_{exp}$) shows slow development at both the high and the low wavenumber range. The case $FN_{exp}$ shows similarly fast convergence as the case $FN_{gauss\_adp}$ for the high wavenumber range, but slower for the lower wavenumbers.
6.5 TURBULENT SCALAR

The previous section was concerned with the generation of turbulent velocity fields. This section explains the generation of additional scalar fields. Scalar fields can be temperature field or pollutant concentration. Turbulent scalar fields can be generated by imposing a mean profile at the inlet and letting the scalar develop into a turbulent field by the turbulent velocity field. This however requires a longer domain and therefore increased computational cost. In order to get a turbulent temperature ABL, it is possible to use a cyclic domain. The energy added at the ground then has to be extracted at the top in order to stabilize the temperature within the boundary layer. This requires careful matching of the ground and top boundary heat fluxes, otherwise the temperature constantly rises. For ABL simulations with horizontally inhomogeneous conditions or including terrain, the cyclic method is not possible. A turbulent scalar inflow generator allows for generation of turbulent temperature boundary layers that match the desired temperature profile. A second application of a turbulent scalar generator is found for pollutant dispersion studies. Figure 6.22 depicts the situation where pollutant from an upstream stack is entrained into a street canyon. Without a turbulent scalar generator, the full domain has to be simulated to study the entrainment. By using a turbulent scalar inflow generator, the domain can be significantly reduced, saving computational time. The plume characteristics have do be defined at the domain inlet, these can be obtained from more simpler models such as a Gaussian plume model or by performing a lower resolution precursor simulation on the full domain. The scalar inflow generator additionally allows to change the plume position and study the effect on the entrainment, without having to reposition the stack and to simulate the entire domain.

A new method is proposed to extend the filtered noise inflow BC to generate any number of additional turbulent scalar fields. The method was developed independently of the recently presented work by Okaze and Mochida (2015), who presented the method for one scalar field. The generation of scalar fields is possible because the filtering step is independent of the correlation step (Cholesky transformation).

This section shows the derivation of the equations for the scalar inflow generator. Subsequently, a simulation that applies the inflow generator is performed. The goal of the simulation is to show that the turbulent
Figure 6.22: Pollutant entrainment into a street canyon, emitted by an upstream stack. The red simulation domain is needed if the full pollutant dispersion is simulated. The blue domain uses a model of the plume and a turbulent inflow generator.

scalar inflow generator is capable of producing turbulent fields with prescribed statistics. To this extent, an upstream line source is computed by using a reference simulation. In the main simulation, a turbulent scalar is generated. This demonstration case can be related to the situation depicted in Fig. 6.22.

6.5.1 Method

The filtered noise method, as defined in Section 6.3, is independent of the dimensionality of the generated variables. It is identical for $n$ additional scalars $\phi_k (k = 1, \ldots, n)$, as it is for just three scalar velocity component fields $u_1, u_2$ and $u_3$.

For each scalar, an additional random field is generated and filtered. The scaling step is then adapted to the new total number of fields. This is possible, as the Cholesky transformation (Eq. 6.5) is defined for an arbitrary number of dimensions. A new covariance matrix is built, using the modified vector
TURBULENT INFLOW GENERATION

\[ u^* = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ \phi_1 \\ \phi_2 \\ \vdots \\ \phi_n \end{bmatrix} \] (6.35)

that includes the scalars. The modified covariance matrix \( R^* \) for \( n+3 \) dimensions reads:

\[ R^* = \text{cov}(u^*, u^*) = \begin{bmatrix} u_1' \ u_1' u_2' \ u_1' u_3' \ u_1' \phi_1' \ \cdots \ u_1' \phi_n' \\ u_2' \ u_2' u_3' \ u_1' \phi_1' \ \cdots \ u_2' \phi_n' \\ u_3' \ u_3' \phi_1' \ \cdots \ u_2' \phi_n' \\ \phi_1' \ \cdots \ 0 \\ \vdots \\ \phi_n' \end{bmatrix} \] (6.36)

The scalar covariances, e.g. \( \phi_1' \phi_2' \), have been set to zero for simplicity, but this is not necessary.

The Cholesky decomposition (Eq. 6.6) is then performed on \( R^* \) according to the algorithm defined in Equations 6.7 and 6.8. For one additional scalar, the decomposition reads:

\[ L_{i,j} = \begin{bmatrix} \sqrt{R_{11}} & 0 & 0 & 0 \\ \frac{R_{12}}{L_{11}} \sqrt{-L_{21}^2 + R_{22}} & 0 & 0 \\ \frac{R_{13}}{L_{11}} \frac{(-L_{21}L_{31} + R_{23})}{L_{22}} & \sqrt{-L_{31}^2 - L_{32}^2 + R_{33}} & 0 \\ \frac{R_{14}}{L_{11}} \frac{(-L_{21}L_{41} + R_{24})}{L_{22}} & \frac{(-L_{31}L_{41} - L_{32}L_{42} + R_{34})}{L_{33}} & \sqrt{-L_{41}^2 - L_{42}^2 - L_{43}^2 + R_{44}} \end{bmatrix} \] (6.37)
where \( R_{ij} \) refers to the components of the modified four dimensional covariance matrix (Eq. 6.36).

### 6.5.2 Computational Setup

A coarse mesh with 723,000 cells was generated, adapted from the flat plate simulation used in Section 6.4. Two scalars are imposed at the inlet (\( x = 0 \) m): one with only a mean profile imposed (non-turbulent) and one using the FilteredNoise inflow generator.

For the turbulent scalar, the FilteredNoise method requires additional vertical profiles of statistics, \( \bar{\phi}(y) \), \( \phi'^2(y) \) and \( \phi'^u_i(y) \). They were sampled from a turbulent reference scalar field, extracted from the case FN\_gauss\_adp at \( x = 0.5 \) m. For both scalars, the same mean inlet profile is used, matching the reference scalar field.

For this demonstration, the same length- and time scales are used for the scalar field as are for the streamwise velocity field \( u_x \).

### 6.5.3 Results

An instantaneous snapshot of the two scalar inflow fields is presented in Figure 6.23. The turbulent scalar field generated with the FilteredNoise method is presented in Fig. 6.23a. The turbulent fluctuations of the scalar are clearly visible, whereas the scalar field without the FilteredNoise inflow generator (Fig. 6.23b) does not show any turbulence.

The effect of scalar turbulence on the downstream evolution is qualitatively shown in Figure 6.24. The filtered noise generated scalar is already turbulent starting at the inlet (Fig. 6.24a), whereas the laminar scalar (Fig. 6.24b) slowly becomes turbulent through the underlying flow field. For both scalars, similar large scale structures are visible further downstream, where the influence of the inflow structures becomes less important and the effect of the local velocity field on the scalar is dominating.

The effect of the turbulence generator becomes clear when investigating the scalar statistics. The variance of the scalar is affected by the inlet far downstream, as visible in Figure 6.25. The variance of the reference scalar field (Fig. 6.25a) shows a decaying plume and the artificially generated turbulent scalar (Fig. 6.25b) is capable of continuing that decay, in contrast to the non-turbulent scalar (Fig. 6.25c) that initially shows no variance.
Figure 6.23: Snapshot of a passive scalar $\phi$ on the inlet patch ($x = 0.0 \, m$), for a) artificially generated fluctuations using the FilteredNoise inflow generator and b) without turbulent inflow generator.

Figure 6.24: Snapshot of a passive scalar $\phi$ on a streamwise vertical plane at $z = 0.0m$, for a) artificially generated fluctuations using the FilteredNoise inflow generator and b) without turbulent inflow generator.

(since only the mean scalar was imposed) and shows a very slow growth further downstream.

The simulation demonstrates that the turbulent scalar FilteredNoise inflow generator is able to reproduce the statistics of a turbulent scalar field. Without the scalar inflow generator (by only imposing a mean scalar
Figure 6.25: Streamwise spatial evolution of the variance of a passive scalar, for a) the reference simulation, b) the artificially generated fluctuations using the FilteredNoise inflow generator and b) without turbulent inflow generator. The dashed line in a) shows the sample location for the statistics.

profile), the turbulence statistics (scalar variance) do not develop to the values of the reference simulation.

6.6 DISCUSSION AND SUMMARY

In this chapter, the choice of the turbulent inflow method used for LES in this thesis was motivated and presented. Due to the high computational efficiency and the high level of control over the generated turbulence, the filtered noise method, originally developed by Klein et al. (2003), was chosen. For this method, a formal way to obtain the filter coefficients was presented. Relations were derived that make it possible to obtain the filter coefficients by specifying the correlation function. Three closed form solutions were presented and validated. In addition, by deriving the streamwise correlation using time series modelling, it was possible to relate the three dimensional approach of Klein et al. (2003) with the simplified two dimensional approach by Xie and Castro (2008). This new insight allows for new possibilities of improving the streamwise correlation by including a Taylor microscale.

The filtered noise inflow generator was implemented in OpenFOAM and simulations were conducted using reference data obtained from the
PIV measurements. The study focused on the filtering, the effect of the imposed temporal and spatial correlation on the development of the flow field. All cases showed a development from the artificial, imposed turbulent fluctuations into coherent structures common to boundary layers (low velocity streaks near the wall, hairpin-like vortices). Convergence of the spatial spectra demonstrated a short development domain. The development section was identified to measures 2-3 times the boundary layer height. It was found, that the shape of the correlation function used for the spatial filtering has an effect on the turbulence development at the wall, even when both the imposed length scale and the imposed Reynolds stresses are the same. An exponential correlation features smaller flow structures, promoting a faster wall turbulence development. The same can be achieved using a Gaussian correlation function when reducing the prescribed length scales at the wall. For the boundary layer turbulence, no effect of the prescribed correlation function on the final shape of the spatial spectra was observed.

A deficiency of the method was observed. The pronounced peak of TKE in the middle of the boundary layer visible in the PIV measurements was generated correctly by the inflow generator at the inlet patch, however a subsequent loss of this peak was observed in the simulation. The loss of TKE for the upper part of the boundary layer was found to happen at two locations: by a loss of vertical and lateral Reynolds stress in the first cell after the inlet and a loss of TKE in the initial section of the domain. The loss within the domain was minimized by choosing a spatial Gaussian correlation function. The loss in the initial cell could not be explained with the parameters that were varied in the presented study. This loss of TKE is also observable in other studies (e.g. Veloudis et al. 2007; Xie and Castro 2008; Okaze and Mochida 2015). Dietzel et al. (2014) and Okaze and Mochida (2015) attributes this loss of TKE to the pressure term having to correct the flow field to make it divergence free. On the other hand, the study presented by Xuan and Iizuka (2013) did not show any improvement when correcting for divergence.

To summarize, the observations made for the simulations lead to the following recommendations:

- If the streamwise integral scales are not well known, imposing too large scales leads to better results than too small scales.
• The lateral and vertical length scales near the wall should be small in order to promote the development of wall turbulence

• Gaussian lateral and vertical correlation shapes lead to the smallest loss in magnitude of the prescribed vertical and lateral fluctuations.

Additionally, it was demonstrated, that the method is not restricted to velocity fields, but can be easily extended to include any number of turbulent scalar fields. This was shown to be useful for an ABL simulation over non-homogeneous terrain with a turbulent temperature boundary layer or to minimize the necessary domain size (and therefore computational cost) for a local scale pollutant entrainment study.
This chapter demonstrates the applicability of the Large Eddy Simulation (LES) based computational model for the urban air for cases of increasing complexity. The model is described in detail in Chapter 5 and should be of high spatial resolution and applicable to an urban setting. This is achieved through a domain size with minimal spatial extent and synthetic inlet turbulence.

A validation and comparison with wind tunnel Particle Image Velocimetry (PIV) measurements of a unit ratio cavity is performed. The validation allows to study the effect of the grid on the predicted dynamics of the shear layer. To separate these effects from the problem of turbulent inflow generation, the validation case is performed with a laminar inflow boundary layer. The experiment (Chapter 4) has shown that the laminar boundary layer case exhibits a turbulent shear layer with the distinct phenomenon of self-sustained oscillations. It is tested if the model can reproduce this.

A second comparison with PIV measurements is made for turbulent inflow boundary layers. Two boundary layers are selected, one where the shear layer (disturbed by upstream turbulence) dominates the cavity flow field and one case where the external turbulence dominates. This shows the ability of artificially generated inflow turbulence to provide realistic turbulent flow conditions.

Finally, the application and potential of the urban air model is demonstrated in the form of a full scale case. The flow field around a three story apartment building in an urban context is simulated under realistic Atmospheric Boundary Layer (ABL) inflow conditions.

The validation is presented in Section 7.1, the computations for the turbulent cavity in Section 7.2, the case study is presented in Section 7.3, followed by a conclusion in Section 7.4.
Figure 7.1: (a) Size of the computational domain and used coordinate system and (b) areas of grid refinement, indicating the levels of refinement (L0-L4).

7.1 CAVITY FLOW VALIDATION

The urban air model, as defined in Chapter 4, is applied to the unity aspect ratio cavity and compared with the wind tunnel measurements (Chapter 4). The PIV case ThinBL features a thin laminar boundary layer (Sec. 4.6).

7.1.1 Computational Setup

With a reference velocity $u_\infty = 3.14 \text{ m/s}$, cavity height $H = 0.1 \text{ m}$ and kinematic viscosity $\nu = 1.55 \times 10^{-5} \text{ m}^2/\text{s}$, the Reynolds number is $Re_H = 20'258$, close to the experiment ($Re_H = 19'260$). The cavity flow was initialized according to the procedure described in Section 5.3.7 and sampling was performed over a duration of 20 seconds.

The boundary conditions used are described in Section 5.3.6. Since the inlet boundary layer is laminar, no turbulent inflow generator had to be used.

7.1.2 Domain and Cases

The cases simulated are listed in Table 7.1. The cases LES_0 to LES_4 use the domain shown in Fig. 7.1a. The base mesh is constructed from cubes
with edge size $\Delta x = 0.004 \text{ m}$ (Level 0). Subsequent mesh refinement is performed by cutting each side in half, resulting in 8 smaller cells with equal size. The areas of mesh refinement are shown in Figure 7.1b, with the levels $L_1-L_4$ denoting each successive refinement. Level 4 is a wall cell refinement. A uniform inflow velocity of 3.14 m/s was imposed at the inlet (block profile).

The inlet boundary of the cases $LES_1_m$ to $LES_3_m$ is adapted to match the experiment: the mean flow profile at the inlet was extracted from the time-averaged PIV measurement and prescribed at the inlet of the computational domain. The domain size was reduced at the inlet from $2H$ to $0.6H$ in order to put the inlet boundary at the position where the PIV inlet profiles were measured. The outlet length was reduced from $2H$ to $1H$ to save computational time. The mesh refinements correspond to the cases $LES_1$ to $LES_3$.

### 7.1.3 Results

The boundary layer profiles just upstream of the cavity ($x = -0.013 \text{ m}$) are shown in Figure 7.2. For the cases with the block inlet profile (Fig. 7.2a), a change with mesh resolution is observed. The boundary layer gets thinner (and the wall gradient higher) the smaller the mesh size. Due to the displacement at the wall, the free stream velocity is higher than the imposed inlet velocity. For the imposed boundary layer cases (Fig.
Figure 7.2: Boundary layer profiles upstream of the cavity for (a) the cases LES_0 to LES_4 and (b) for the cases LES_1_m to LES_3_m, compared to PIV measurements.

7.2b), all cases match the imposed PIV profile well and do not develop significantly. The coarser case LES_1_m shows a slight deviation.

For the cases with the block inlet profile, the resulting velocity fields in the cavity are presented in Figure 7.3. The two coarse mesh cases LES_0 and LES_1 show a significantly lower mean flow velocity in the cavity than the PIV measurement. Additionally, no vortex shedding in the shear layer is visible in the instantaneous vertical velocity, resulting in virtually zero Turbulent Kinetic Energy (TKE) in the shear layer and cavity. The finer mesh cases show a better match for the mean flow field with only small differences. The shear layer dynamics of the case LES_2 match the experiment best, specifically the shape of the shear layer TKE. The cases LES_3 and LES_4 are similar to each other, and show a significantly larger area and magnitude of TKE. The instantaneous vertical velocity shows a different pattern of vortices than the case LES_2, spaced closer together.

For the cases with the imposed boundary layer, Figure 7.4 shows the resulting flow fields. Similar to the case LES_1, the coarse mesh case
Figure 7.3: Comparison of mean velocity magnitude, instantaneous vertical velocity and turbulent kinetic energy of the cases LES_0 to LES_4 with PIV measurements.
Figure 7.4: Comparison of mean velocity magnitude, instantaneous vertical velocity and turbulent kinetic energy of the cases LES_1_m to LES_3_m with PIV measurements.

LES_1_m shows a low cavity flow and no TKE. The refined cases, LES_2_m and LES_3_m, both show similar velocity fields as the PIV measurements, and the shape of the TKE is predicted correctly. The magnitude of TKE in the shear layer is however lower for the coarser case. The vortices in the shear layer (pattern in the instantaneous vertical velocity fields) seem to match well.

Figure 7.5 shows profiles of mean velocity, TKE and shear stress for a horizontal and a vertical line through the cavity center. The mean
flow velocities $\overline{u}_x$ and $\overline{u}_y$ for the coarse mesh case LES_1_m is severely underpredicted and only a very small amount of TKE can be observed at the left (Fig. 7.5a) and lower cavity wall (Fig. 7.5b). For the cases LES_2_m and LES_3_m, the shear layer TKE and shear stress $\overline{u}_1'\overline{u}_2'$ is well predicted, however the case LES_3_m shows slightly more shear stress than the PIV measurement. It is likely that the lower resolution of the PIV measurements smoothen out the peak of TKE and $\overline{u}_1'\overline{u}_2'$, as it averages over the interrogation area. The mean flow velocities are well predicted by both LES_2_m and LES_3_m. LES_3_m performs slightly better. A mesh dependency for $\overline{u}_y$ can be observed at the right wall (Fig. 7.5b), with a steeper wall gradient for the higher resolution case. TKE and $\overline{u}_1'\overline{u}_2'$ at the right wall are also underpredicted.

To further investigate the shear layer, the spectra of vertical velocity are compared for a location at the center of the cavity top ($x = 0.05$ m and $y = 0.0$ m) and presented in Figure 7.6. Only the cases that show a turbulent shear layer are considered. A comparison between the block profile cases and PIV (Fig. 7.6a) shows clear differences in both the location of the peaks and the overall magnitude. The difference between the two LES cases cannot be explained with a difference in mesh resolution only. The different behaviour of the shear layer is presumably caused by the different inflow profiles (Fig. 7.2a). The boundary layer profile of the case LES_2 shows a better match to the PIV profile than the case LES_3 and the flow fields are closer to the measurements as well. As the boundary layer thickness is an important parameter in determining the mode of the self sustained oscillations (Knisely and Rockwell 1982), a switch between mode $m=2$ and $m=3$ could explain the differences between the two cases.

The spectra for the imposed boundary layer cases are similar to PIV (Fig. 7.6b), the fundamental frequency and the frequencies of the harmonics are captured well. The grid influence is clearly visible through the loss of energy towards higher frequencies, but this does not change the location of the frequency peaks. For the comparison with PIV it has to be considered that the measurements also do not capture the higher frequency spectrum well due to the added PIV noise. Therefore a comparison of the high frequency dissipation cannot be made. However, the case LES_3_m shows good agreement in the spectrum up to 70-80 Hz, and the important dynamics of the shear layer are captured by both simulations.
Figure 7.5: Profiles of mean velocity, TKE and shear stress $u'_1u'_2$ for the cases LES_1_m, LES_2_m and LES_3_m compared with PIV, through the cavity center for a) a vertical line at $x = 0.05$ m and b) a horizontal line at $y = -0.05$ m.
Figure 7.6: Power spectral density $\Phi$ of the vertical velocity component $u_y$, for the cases without controlled inlet ($LES_2$ and $LES_3$, left) and with controlled inlet ($LES_2_m$ and $LES_3_m$, right) compared to PIV measurements (case ThinBL), evaluated in the shear layer at $x = 0.05$ m and $y = 0.0$ m.

7.1.4 Conclusion

A validation case has been designed in order to test the simulation model without the added complexity of turbulent inflow generation. The LES model is able to capture the cavity flow field and the dynamics of shear layer correctly, if the mesh resolution is sufficiently fine. The validation only works if the conditions of the experiment are matched, which is not the case if only a block profile is imposed at the domain inlet: a series of grid refinements have revealed the sensitivity of the validation case with respect to the inlet boundary layer and therefore the shear layer dynamics. With the PIV inlet profile imposed, a comparison to the measurements could be achieved. It was shown that a too coarse mesh resolution in the shear layer region suppresses the development of shear layer oscillations, leading to a wrong prediction of the flow field. Both mesh resolutions of the cases $LES_2_m$ and $LES_3_m$ predicted the correct oscillation frequencies of the shear layer and show a good prediction of the cavity flow field. The mesh refinement in the shear layer for the case $LES_3_m$ resulted in the best agreement with the experiment, most notably by the higher TKE in the shear layer.
The comparison of flow profiles did show a dependency of the right wall boundary layer with the mesh resolution. It was demonstrated, that resolving the wall further (case $LES_4$) did not influence the overall flow field significantly, leading to the assumption that the flow field in the cavity is primarily governed by the larger structures produced in the shear layer and the smaller structures generated by the cavity wall boundary layer have a smaller influence on the cavity flow field. Additionally, wall refinement doubled the cell count, leading to a much higher computational cost.

### 7.2 Turbulent street canyon

In this section, the application of the full urban air model is demonstrated, using the cavity case with a turbulent inlet boundary layer. Simulations for two different boundary layers extracted from wind tunnel measurements were conducted: a 10 cm moderately turbulent boundary layer and a 15 cm strongly turbulent boundary layer ($PIV$ cases $10m$ and $15w$, Section 4.2). For the $10m$ boundary layer, the input-output behaviour of the FilteredNoise Boundary Condition (BC) was investigated in detail in the Chapter 6.

#### 7.2.1 Computational Setup

The computational domain and mesh refinement is shown in Figure 7.7. The same outlet fetch distance was used as in validation study (cases $LES_1.m$ to $LES_3.m$). The inlet fetch was kept short to save computational time and was set to 0.2 m. This distance was chosen based on the studies on the evolution of artificial turbulence (Chapter 6) that showed development of wall turbulence after 0.2 m. The mesh refinement levels from Table 7.1 are used, with the finest resolution in the domain being level 2. The shear layer was not further refined, as was done for the thin laminar boundary layer case $LES_3.m$, because a less steep velocity gradient is observed in the $PIV$ measurements for the turbulent cases (e.g. $PIV$ case $10m$ or $15w$). The refinement border for the levels 1 and 2 are raised to include the full turbulent boundary layer ($\approx 0.1$ m). The wall was refined further in wall normal direction as displayed in the inset in Fig. 7.7b ($L3$). The resulting mesh counts $7'348'160$ cells.
Figure 7.7: (a) Size of the computational domain and used coordinate system and (b) areas of grid refinement, indicating the levels of refinement (Lo-L3).

For the inlet boundary condition, the turbulent inflow generator FilteredNoise was used (Chapter 6). An exponential Autocorrelation Function (ACF) and Gaussian spatial correlation was used.

7.2.1.1 Cases

The cases simulated for the 10m and 15w boundary layer are listed in Table 7.2.

For the 10m boundary layer, two additional cases are computed: one with a laminar boundary layer only (LES_10m_lam), and one with adapted lateral and vertical length scales (LES_10m_adp), sampled at a location 5H downstream of flat plate domain (from the case FN_gauss_adp in Section 6.4). The inflow data for the case LES_10m is shown in Figure 6.8, for the case LES_10m_adp in Figure 6.9. The inflow data for the case LES_15w is shown in Figure 7.8.

7.2.2 Flow Fields

The resulting flow fields of the 10m cases are presented in Figure 7.9. The case with laminar inflow (LES_10m_lam), not surprisingly, is unable to reproduce the PIV measurement. Even though the mean inlet boundary layer matches the experiment, the velocity in the cavity is significantly too low. The shape of the shear layer TKE is similar to the laminar thin
Table 7.2: Cases for the LES of the turbulent inflow cavity

<table>
<thead>
<tr>
<th>Case</th>
<th>PIV Case</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>LES_10m</td>
<td>10m</td>
<td>turbulent inflow</td>
</tr>
<tr>
<td>LES_10m_adp</td>
<td>10m</td>
<td>turbulent, adapted lateral and vertical scales</td>
</tr>
<tr>
<td>LES_10m_lam</td>
<td>10m</td>
<td>laminar inflow</td>
</tr>
<tr>
<td>LES_15w</td>
<td>15w</td>
<td>turbulent inflow</td>
</tr>
</tbody>
</table>
boundary layer case LES\_2\_m (Section 7.1), suggesting that the shear layer is unstable and weak shear layer oscillations exist. A turbulent shear layer as observed in the PIV measurement however does not develop.

For both turbulent inflow cases (LES\_10m and LES\_10m\_adp), the mean flow field topology matches the measurement, with a slight underprediction of the velocity magnitude. This is visible in the vertical and horizontal profiles (Fig. 7.10). The fields of TKE correspond with the simulation, and are only underpredicted slightly within the cavity, especially at the cavity floor.

Differences are noticeable in the fields of TKE production (Fig. 7.9) in the shear layer. The PIV case shows a strong production in the first half of the shear layer and then stabilizes at a lower level towards the second half. The simulation with the adapted length scales (LES\_10m\_adp) captures this trend and shows a better match with the measurement than the case with the simplified length scale profiles. The shear layer of the PIV case 10m is similar to the case 5m (investigated in Section 4.6). In Figure 4.30 it was shown that the initial peak of $P_{\text{TKE}}$ is caused by the rise of the shear stress $-u_1^' u_2^'$. The better match of the adapted case can therefore be related to the better prediction of the shear stress, as presented in Figure 7.10a and 7.11. As the mesh is identical for both simulations, this suggests that the case 10m is sensitive to the near wall turbulent structures present upstream of the cavity.

For the highly turbulent inflow boundary layer (15w), the comparison of the cavity flow fields between the computation and the PIV measurement is shown in Figure 7.12. Given that the LES inflow turbulence was generated artificially, the mean flow field and TKE in the shear layer and cavity is predicted remarkably well. The location and shape of the TKE production at the beginning of the shear layer is also captured well by the simulation. Differences exist in the velocity near the bottom and right wall, as apparent in Figure 7.13a and 7.13b, where LES slightly underestimates the velocity. TKE within the cavity is overall lower than the measurement, in contrast to the shear layer, where TKE is captured well. The trend of the shear stress is generally captured well.

The PIV measurement 15w shows distinct, irregular large sweep events in the cavity shear layer (Figure 4.33). For the LES, the shear stress was sampled at nine points (at $y = 2$ mm, regularly spaced between $x = 0.01$ m and $x = 0.09$ m) and was spatially averaged over these points to recreate the PIV result. The spatially averaged shear stress, decomposed into
Figure 7.9: Comparison of mean velocity magnitude, turbulent kinetic energy and production of TKE of the cases LES_10m_lam, LES_10m and LES_10m_adp with PIV measurements.
Figure 7.10: Profiles of mean velocity, TKE and shear stress $u'_1 u'_2$ for the cases LES_10m_lam, LES_10m and LES_10m_adp compared with PIV, through the cavity center for a) a vertical line at $x = 0.05 \, m$ and b) a horizontal line at $y = -0.05 \, m$. 

7.2 TURBULENT STREET CANYON
the four quadrants, is presented in Figure 7.14 for the same duration ($T = 5.25$ s) as the measurement. Compared to the measurement, the simulation also shows large sweep events with similar magnitude. For the smaller events, the simulations shows more ejection whereas the measurement shows more sweep events.

7.2.3 Flushing

For the $10m$ cases and the $15w$ case, a scalar flushing was simulated. Starting from a statistically stationary flow field, the cavity was initially ($t = 0$) filled with a scalar value of 1.0. The volume integral of the scalar within the cavity was sampled over time, until 5\% of the initial value was reached. No sources were used and only one realization per case was computed. One realization was deemed sufficient, as the volume integral acts like a lateral averaging of several independent vertical planes. This assumption is justified since the cavity geometry is two dimensional and the lateral domain size (0.16 m) is larger than the lateral integral length scales of the flow ($\approx 0.02$ m for the case $15w$ and $\approx 0.04$ m for the case $15w$).

The resulting scalar decay is presented in Figure 7.15. The vertical axis is logarithmic, revealing an exponential decay (linear lines in semilog plot). The following observations can be made: The lowest rate of scalar removal (shallowest slope of the lines) is shown by the case $\text{LES}_{10m}\text{laminar}$, resulting in the longest flushing time. When adding turbulence (case $\text{LES}_{10m}$), the flushing time decreases significantly (by
Approximately 45%. The adapted case \textit{LES\textsubscript{10m\_adp}} shows only a small decrease in flushing time over the case \textit{LES\textsubscript{10m}}. The strongly turbulent case \textit{15w} shows the fastest removal. The laminar, thin boundary layer inflow case \textit{LES\textsubscript{3m}} shows a flushing time between the case \textit{10w} and \textit{15w}.

For all cases, two distinct removal rates can be observed. They are especially pronounced for the cases with laminar inflow boundary layer (\textit{LES\textsubscript{3m}} and \textit{LES\textsubscript{10m\_lam}}), and are least apparent for the case \textit{LES\textsubscript{15w}}. An initial, steeper slope is followed by a shallower slope. During the initial phase, a large amount of scalar is removed (up to \(\approx 50\%\)). A similar phenomena of an early and late time distinct removal rate was observed by McCoy et al. (2005), who simulated a scalar flushing between two walls in a channel using \textit{LES}. It is presumed that the scalar near the shear layer and walls are removed first from the cavity, while scalar is being trapped in the center of the cavity vortex. In the later stage, the scalar from the vortex is being transported outwards, where it can subsequently be
Figure 7.13: Profiles of mean velocity, TKE and shear stress $u'_1u'_2$ for the case $LES_{15w}$ compared with PIV, through the cavity center for a) a vertical line at $x = 0.05$ m and b) a horizontal line at $y = -0.05$ m. Every other PIV sample is displayed for clarity.
Figure 7.14: Temporal evolution of the spatially averaged shear stress $\overline{u_1' u_2'}$ along the cavity top at $y = 2 \text{ mm}$. Colored lines show the contribution of the 4 quadrants, and the colored ares show the difference in magnitude of sweep and ejection.

removed through the shear layer. This model is supported by the analysis of the scalar separately for the vortex and the outer area. For the case LES_3m, time resolved data on a streamwise vertical, a two dimensional plane was sampled in the cavity during the flushing. Figure 7.16 shows the decay of the scalar in a box centered in the cavity ($dx = dy = 0.070 \text{ m}$) and in the outer area surrounding the inner box. Both areas initially hold 50\% of the scalar. The removal from the outer area is initially faster than from the inner area, up to about $t = 2 \text{ s}$. The snapshot of the scalar field at $t = 1.001 \text{ s}$ clearly shows the higher concentration in the inner box surrounding the vortex in comparison to the outer area. Figure 7.17 shows the same for the case LES_15w. The faster removal from the outer area is observed until about $t = 1 \text{ s}$. A concentration difference between the vortex (inner area) and outer area is still visible, but not as pronounced as for the case LES_3m.
Figure 7.15: Decay of the scalar $\phi$ (integrated over the cavity volume and normalized by initial value $\phi_0$), until 5% of the initial value. The black dashed lines approximate the slope at the beginning.

This can explain the similarities between the laminar case $\text{LES}_3m$ and the turbulent case $\text{LES}_{15}w$. Both show a similar initial removal rate. The late stage removal of the laminar case is however slower. The case $\text{LES}_3m$ is as efficient in removing the scalar of the scalar from the cavity as the case $\text{LES}_{15}w$, due to the higher shear layer velocity. In the late stage however, the case $\text{LES}_{15}w$ is more efficient in removing the scalar, due to the high degree of turbulence (and therefore mixing) within the cavity. The mixing of fresh air with the vortex is visible in the snapshot of $\phi$ in Figure 7.17.

7.2.4 Conclusion

The LES model with a turbulent inflow generator was applied to the cavity case, enabling a direct comparison to PIV measurements. The computational domain and the mesh was created according to the findings of the grid study for the laminar case. Instationary inflow conditions were provided through the filtered noise method. The comparison of the average flow field (mean flow and turbulent statistics) showed very
Figure 7.16: Decay of scalar $\phi$ in the cavity, integrated on a vertical streamwise plane for the case LES$_{3m}$, until 5% of the initial value.

Figure 7.17: Decay of scalar $\phi$ in the cavity, integrated on a vertical streamwise plane for the case LES$_{15w}$, until 5% of the initial value.

good agreement with the PIV measurements. The case 10$m$ showed some sensitivity with the near wall turbulence upstream of the cavity. With adapted, smaller length scales at the wall, the inflow generator could provide satisfactory turbulence in order to show the same shear
layer phenomena as observed in the PIV measurements. The case showed the ability to generate highly turbulent inflow. The external flow turbulence is the main influence for the cavity flow field for this case. The inflow generator proved capable of generating large scale flow structures, showing similar sweep events as the measurement. With the comparison to PIV giving confidence in the predicted flow fields, the ability of the cavity to remove a passive scalar was investigated using LES. It was found that the removal is governed by two processes: the transport of scalar from the center of the cavity (vortex) to the outer area, and the subsequent removal through the shear layer.

7.3 CASE STUDY

This section presents a case study for an apartment building in an urban environment. The aim of this study is to show that the urban air model (especially the turbulent inflow generator) can be applied at full scale. The case study uses a domain of small spatial extent where a rough inlet ABL, an Urban Boundary Layer (UBL), is provided by the inflow generator.

A three story apartment building is used for the case study. The geometry is simplified for the simulation by removing all surface detail and construction elements, as visualized in Figure 7.18. The building is composed of two blocks: a smaller, L-shaped block and an elongated block at an angle of 30 degrees. Individual floor to floor heights pertain to 3 meters, with an overall building height of \( H = 9 \) m. The top floor consists of two units which are set back compared to lower floors. This makes room for large terraces at the longitudinal side of the building as well as between the two units, creating a lower roof area in the middle of the building. The building is aligned west to east and is loosely based on the existing Überbauung Pfaffenbühl (Müller & Messerli AG 2006).

To visualize the turbulent structures created by the inflow generator, only the apartment building is resolved and is placed on a flat surface. This case is denoted isolated. Isolated refers to the building and not the imposed UBL. For a second case, three additional buildings are resolved. The role of these buildings are to support the inflow generator by additional geometrically generated turbulence. This turbulence should show realistic coherent structures, as present in an urban environment. The cases is therefore denoted urban.
7.3 CASE STUDY

Figure 7.18: Visualization of the apartment building (red) for the case urban, surrounded by three generic buildings (white).

7.3.1 Inlet Profiles

The FilteredNoise inlet boundary is used to generate turbulent inflow conditions. The required inlet statistics for a neutral urban UBL were extracted from an existing precursor simulation performed using the ABLSolver code (SOWFA 2015). The solver uses horizontally cyclic boundary conditions and a wall shear stress model. A surface roughness of $z_0 = 0.5\, m$ was used, corresponding to very rough according to the classification of Wieringa (1992). The domain is a box with $3\, km \times 3\, km$ horizontal extent and vertical extent of $1.6\, km$. This resulted in a turbulent boundary layer with a height of $\delta \approx 800\, m$. Velocity point probes were sampled on a vertical line in the center of the domain to compute the integral length scale. Snapshots of the velocity field were sampled on two vertical planes (streamwise and spanwise) in order to compute the lateral and vertical length scales. The extracted profiles are presented in Figure 7.19.

The linearly varying length scale profiles generate small turbulent structures near the ground and large scale structures the upper part of the boundary layer. The generated inflow turbulence has an exponential auto-correlation function and a Gaussian spatial correlation function (vertical and lateral).
Figure 7.19: Boundary layer input data for the FilteredNoise BC, with mean velocity $\bar{u}_x$, Reynolds stresses $R_{ij}$ and integral length scales $L_{ij}$

7.3.2 Computational Setup

The same meshing strategy as for the validation cases is used. A coarse base mesh (level 0) is generated, with cubical cells having a resolution of $2m \times 2m \times 2m$. Subsequent refinement is performed on selected areas, as presented in Figure 7.20. The refinement boxes are centered around the building and oriented in streamwise direction (x-axis). The finest cells (refinement level 4) are a wall refinement layer at the building walls ($0.125m \times 0.125m \times 0.125m$) and is four cells thick ($0.5m$ away from the wall). The overall extent of the domain is $100m$ vertical, $100m$ lateral and $90m$ streamwise. The resulting mesh size is $4'165'747$ cells.

The selected mesh resolution is put in context with the previous simulations. The ratio $L/\Delta x$, where $L$ is the integral length scale and $\Delta x$ is the cell size, can be understood as the number of points that are used to resolve an energy containing eddy. The largest length scale for the case LES_15w was $L = 0.05m$ and with the mesh resolution in the boundary layer (not at the wall) $\Delta x = 0.001m$, this results in $L/\Delta x = 50$. The cell size in the lower part of the ABL for this simulation is $\Delta x = 0.25m$ and
with the length scale at $y = 2\, m$ of $L = 5\, m$ and $L = 19\, m$ at rooftop level results in ratios of 20 and 76 respectively. This means that the large scales are similarly well resolved as for the cavity case. The Reynolds numbers however differ by a factor of approximately 90 (full scale building height of 9 m to model scale height 0.1 m of the cavity), meaning that less of the small scales are resolved.

A second mesh with the three additional surrounding buildings was generated. Since the lateral buildings are extending into coarse mesh regions, the walls were further refined to level 2 and the edges to level 3. The total mesh size is 4'388'939 cells.

For each case, a simulation for one wind direction was run: The inlet flow direction is the positive x-axis (west wind, Fig. 7.21). The choice of wind direction was made in order to have at least one building directly upstream up the apartment building. The timestep is set to $\Delta t = 0.01\, s$, resulting in a mean Courant number of $Co = 0.036$ and a maximum of $Co = 0.57$. The initial flow field was set using a coarse mesh laminar RANS solution. The flow was initialized for 360 seconds. The subsequent sampling of statistics was conducted over a period of 2250 seconds. With an integral timescale at mid-building height of $T_1 \approx 8\, s$ (from the inlet UBL, Fig. 7.19), this results in 140 independent samples. This translates into a standard error of the mean of $SE_{\bar{u}_1} = 4.5\%$. 

**Figure 7.20:** Extent of the computational domain and mesh refinement levels.
Results

Results of the simulations are presented for the turbulent flow field generated by the inflow generator and for the effect of surrounding buildings on the flow field.

7.3.3 Turbulent Flow Field

The inflow statistics generated by the FilteredNoise inflow boundary condition is presented in Figure 7.22. The mean velocity profile matches the imposed UBL profile well for the lower 50 m of the boundary layer and shows a slight velocity deficit in the upper half. The Reynolds stresses for the vertical and lateral components ($u_2^2u_2^2$ and $u_3^2u_3^2$) follow the imposed profiles (within statistical accuracy). The horizontal component $u_1^1u_1^1$ however shows large differences. The maximum value is increased by about 20%, with the position of the maximum located at a slightly lower height. The upper boundary layer values are severely too low. This also impacts
Figure 7.22: Vertical profiles of horizontal velocity $u_x$ and Reynolds stresses $u'_i u'_j$ at the inlet ($x = -50$ m), compared to the input dataset (dashed).

The discrepancy between the actual and the imposed Reynolds stress $u'_1 u'_1$ is further investigated. As the statistics are sampled on the inlet patch and not inside the domain, an influence of the flow computation on the input statistics can be excluded. A plausible error source is the mass flow correction implemented in the FilteredNoise BC code. The mass flow correction ensures that the mass flow into the domain is constant, meaning the spatial average of the patch normal fluctuating inlet velocity is zero: $\langle u'_1 \rangle = 0$ m/s. For each time $t$, the inlet normal velocity fluctuations $u'_1$ are summed over all $N$ inlet faces to obtain the volume flow difference $\Delta \dot{V}(t)$:

$$\Delta \dot{V}(t) = \sum_{k=0}^{N} u'_1[k](t) A. \quad (7.1)$$
The flow velocity at each inlet face is subsequently corrected using:

$$u'_{1, corr}[k](t) = u'_1[k](t) - \frac{\Delta \dot{V}(t)}{A_{tot}} u_{corr}(t),$$  \hspace{1cm} (7.2)

where the term $\Delta \dot{V}/A_{tot}$ is the mass flow correction velocity $u_{corr}(t)$. As $u_{corr}(t)$ is constant over the inlet patch, the correction can be interpreted as a constant shift of the generated fluctuating velocity field $u'_1$ by $u_{corr}(t)$ (therefore fulfilling the requirement $\langle u'_{1, corr}[k](t) \rangle = 0 \text{m/s}$). A time trace of this correction velocity is presented in Figure 7.23 as a percentage of the mean UBL velocity at the top of the domain. Large corrections (as high as 20%) can be observed, showing a long correlation over time. In addition, the large corrections are superimposed by small ($\approx 1\%$), short time fluctuations. The small corrections are consistent to what can be expected.

The input dataset (Fig. 7.19) shows that the generated flow structures get larger with increasing height, the vertical and lateral length scales of the streamwise velocity reach 92 m and 55 m respectively. As the domain size is only 100 m $\times$ 100 m, these large structures at the top of the domain result in a large imbalance of the instantaneous mass flow and lead to a correspondingly large correction velocity $u_{corr}(t)$. This correction essentially negates the imposed $u'_1 u'_1$ Reynolds stress component at larger heights, as visible in Fig. 7.22. However, the correction velocity has a variance of 0.098 m$^2$/s$^2$ (Fig. 7.23) that is being added to the final velocity. This is clearly visible as a constant shift of the $u'_1 u'_1$ profile by 0.098 m$^2$/s$^2$. 

**Figure 7.23:** Time trace of the mass flow correction term $u_{corr}$. 

![](image)
7.3 Case Study

Figure 7.24: Horizontal planes of instantaneous fluctuating streamwise velocity $u'_x$ at $y = 2\, m$ and $y = 7\, m$ for the case isolated.

in Figure 7.22. This explains the increased maximum at lower heights (where the length scales are still small) and the non-zero stress towards the ground. Additionally, this explains why only the magnitude of the streamwise fluctuations are affected. The mean flow velocity and the length scales of the generated turbulent structures are not changed by the mass flow correction.

For the isolated case, the instantaneous turbulent velocity field for the streamwise component $u'_x$ is visualized in Figure 7.24 for two horizontal planes (A-A and B-B, Fig. 7.21). The plane A-A at $y = 2\, m$ shows development of turbulent structures immediately after the inlet. Streak-like structures can be observed in both the refined-mesh central region and the coarser outer region. This shows that the smaller scale turbulent structures are not negatively affected by the mass flow correction. Additionally, it is speculated that the small scale fluctuations added by the correction near the wall might promote a faster transition from artificial turbulence to more realistic turbulent structures. At the higher elevation (plane B-B, $y = 7m$), larger structures are generated. These structures
are observed to persist over a longer distance, until the small scale turbulence generated by the building becomes dominant. The building turbulence seems to have a large lateral area of influence, as small scale structures are visible up to the refinement borders \((y \approx \pm 25 \text{ m})\). Figure 7.25 shows the instantaneous velocity field on a vertical streamwise plane (C-C). Downstream of the inlet and near the ground, all components show development of smaller scale turbulent structures. This is consistent with Figure 7.24. Additionally, the lateral component \(u_z\) shows upwards tilted flow structures typical for boundary layers. In the upper part of the domain \((y > 5 \text{ m})\), the horizontal component initially shows a lack of turbulent structure immediately after the inlet. The \(u_y\) and \(u_z\) fields show artificial turbulence structures, visible as vertical, slightly forward
tilted patterns. Further downstream, all components show large scale variations. The artificial patterns disappear around $x \approx -30 \, m$, before reaching the building. From the horizontal velocity field, two clear flow detachments are visible. One located at the downstream edge of top floor of the first block and the second at the downstream edge of the second block. The wake zone behind the building is clearly visible. Large lateral velocities on the roof (positive) and in the wake (negative) show the three dimensionality of the flow field.

7.3.3.2  Isolated vs. Urban

The investigation of the inlet statistics and the instantaneous flow field of the isolated case have shown that despite the deficiencies in the reproduction of the Reynolds stress $\overline{u'_1u'_1}$, the FilteredNoise BC succeeds in generating a suitable turbulent flow field. The local wind environment around the apartment building is further investigated for the two cases isolated and urban.

Figure 7.26 visualizes the three dimensional coherent structures using the Q-criterion. The isolated case (Fig. 7.26a) shows a horseshoe vortex around the upstream side of the building. The vortex is interrupted and disturbed by other smaller vortices and is therefore not stationary. Additional vortex structures are being shed from all upstream edges of the building. The urban case (Fig. 7.26b) shows a large increase of coherent structures at the north-west side. These vortices originate from the wake flow of the upstream building and show a high vertical velocity. A horseshoe vortex can still be identified on the south-west side of the building. Additionally, a significantly higher occurrence of vortex structures is visible in the roof area between the two top floor blocks.

Figure 7.27 shows the instantaneous vertical fluctuating velocity for a horizontal plane at $y = 2 \, m$, comparing the isolated and the urban case. The isolated case clearly shows the horseshoe vortex and the turbulent wake flow on the south side. The urban case shows similar flow features at the south side of the apartment building as the isolated case, however the flow field upstream and north of the apartment building is greatly influence by the wake of the upstream building. The upstream building shows a strong horseshoe vortex, extending around the south side. On the north side, the vortex is disturbed by the adjacent building. This additional building (in the north) also shows indications of a horseshoe vortex. This vortex
Figure 7.26: Iso-surfaces of positive instantaneous $Q$, colored by vertical velocity $u_y$ for the case (a) isolated and (b) urban (top view).

However does not warp around the whole building, but seems to stop at around $z = -30$ m. This is where the mesh refinement border is located. The mesh at the coarser levels is therefore clearly too coarse to resolve important turbulent features. This can also be seen for the added building in the south. This building is almost entirely in the coarse mesh region (Fig. 7.20, cell size 1 m) and also does not seem to generate any turbulence flow features. Since these structures would propagate downstream and therefore not directly interact with the apartment building, the effect on the result is deemed small.

The flow statistics, mean flow and TKE, are presented in Figure 7.28 and 7.29. Both figures each show the flow on two different horizontal planes, A-A ($y = 2$ m) and B-B ($y = 7$ m), with a side-by-side comparison of the two cases isolated and urban. Figure 7.28 shows the mean flow velocity, with the flow direction visualized by streamlines. For both cases,
the flow velocity is higher on the upper plane, due to the imposed UBL profile. First we discuss the $y = 2\,m$ plane. For the *isolated* case, the flow accelerates around the building (due to the blockage of the building) and a higher flow velocity is visible to the north and south of the building. For the *urban* case, the additional buildings amplify this effect by creating a channeling and further accelerate the flow. This channeling is particularly visible between the apartment building and the southern building. The corner flow detachment at both buildings further aide to that effect. Figure 7.29 shows that the TKE does not increase significantly between the two buildings, making this mostly a mean flow effect. The effect of the upstream building on the flow field of the apartment building is most noticeable on the north side of the apartment building. The wind velocity at the north-eastern corner is significantly higher than for the *isolated* case. This results in a larger separation zone at the northern facade, but also to higher velocities after the re-attachment further downstream. TKE is significantly larger in the entire area between the apartment building, the upstream building and the northern building. This increase can be
Figure 7.28: Streamlines of mean horizontal velocity on two horizontal planes (A-A and B-B), for the cases isolated and urban. The background is colored by the magnitude of the mean velocity.

attributed to the wake flows of the upstream buildings. At the higher plane \((y = 7\ m)\), the flow field and differences between the cases are largely similar to the lower height: the additional buildings generally increase the flow velocity and increase TKE. Most noticeable differences are visible on the north side terraces of the angled block. The wind velocities and TKE are higher compared to the isolated case. A significant difference can also be observed on the lower roof level between the two blocks. The flow in the wake region downstream of the building does not seem to be influenced much by the upstream buildings.

Figure 7.30 presents radial histograms in the form of wind rose plots for three probe locations. The opening angle of one bar is 8 degrees. The probes 1 and 2 are located at \(y = 2\ m\) above the ground, each at distance
of 0.5 m from the edge and 0.5 m away from the wall. For the \textit{isolated} case, probe 1 shows a strong directionality from the south (parallel to the wall), with \approx 40\% of the samples in a narrow angle (23 degrees) and with a broad distribution of velocities. The same location for the \textit{urban} case, shows an even narrower range of directions and the distribution shifted to higher velocities. This is consistent to what has been observed in Figure 7.28. The probe at location 2 shows a different wind rose between the two cases. The \textit{isolated} case shows a wide distribution of directions centered around W-S-W, with high peak velocities. With the exception of N-W, all other wind directions also occur, with slow velocities between 0 and to 1.0 m/s). This pattern suggests that the probe is located in the shear layer generated by the detachment at the building edge. The shear layer is turbulent and not stationary, as the probe location is sometimes inside

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure7.29.png}
\caption{TKE on two horizontal planes (A-A and B-B), for the cases \textit{isolated} and \textit{urban}).}
\end{figure}
Figure 7.30: Wind rose plots for three selected locations, for the cases isolated and urban.
the reversed flow zone (low velocities, direction east) and sometimes in the high speed corner flow (W-S-W). In contrast, the urban case shows mostly reverse flow at the same location. The larger separation zone is clearly visible in the mean flow field (Fig. 7.28). The more rare but higher velocities (from the W-S-W) can be attributed to vortical structures from the upstream building, disturbing the reversed flow zone occasionally. These vortices are visualized in Figure 7.26. Probe 3 is located on the roof area between the two building blocks. Consistent with the observations for the mean flow field and the TKE, the wind rose for the urban case shows a broader distribution and slightly higher peak velocities. For the isolated case, the main wind direction is aligned with the angle of the second block. This is not the case for the urban case, the flow direction is however still predominantly from a similar direction (W-N-W).

### 7.3.4 Discussion and Summary

Flow simulations around a three story apartment building applied the urban air model for complex geometries. The simulation was conducted at full scale, using an artificially generated turbulent boundary layer (UBL). The domain size was purposely kept small to reduce the computational time and this has revealed problems with the generation of turbulence in the upper part of the ABL. The integral length scale at the top of the domain is in the order of the domain size. The mass flow correction at the boundary leads to a modification of the magnitude of the generated streamwise fluctuations. The results did not show any problems, probably because the fluctuations are only reduced in the upper part of the ABL and not at building height and the other flow properties (mean flow velocity, length scales) are not affected. The spatial extent of the domain could easily be expanded in the vertical and lateral direction with coarse sized cells, in order to alleviate the mass flow correction problems. This can be done without much additional computational cost. A turbulent flow field develops after the inlet, showing streak-like turbulent features at the wall. The chosen mesh resolution was able to resolve typical turbulent flow features, such as horseshoe vortices or vortex shedding at building corners. The computation was performed on a relatively small mesh (4.2 mio cells), available computational resources would allow for a finer mesh. This study did not address the quality of the mesh
in detail, however a comparison with the turbulent length scales of the cavity simulations showed that the mesh resolves the large scales similarly well as the validated cavity simulations. Ideally a comparison with wind tunnel data should be made. An additional resolved building placed upstream generated coherent structures through vortex shedding that interact with the apartment building. Two additional resolved buildings placed laterally mostly influence the mean flow field. The mean flow velocity and the TKE was significantly influenced by these three additional buildings. Besides flow statistics such as the mean flow, or TKE, wind rose plots are used for selected locations. This type of plot additionally shows the variation of (horizontal) wind direction. The surrounding buildings are therefore needed and most probably cannot be replaced with an artificial inflow generator. They do however not lead to an increased computational cost.

7.4 CONCLUSION

In this chapter, a validation of the LES based urban air model was presented and an application to an urban case demonstrated.

The validation was performed in two steps for a cavity type flow, without and with inflow turbulence. Wind tunnel measurements (time resolved stereo PIV) served as a reference. The case without inflow turbulence has demonstrated the ability of the model to capture and replicate dynamic shear layer phenomena observed in the measurements. It was also revealed, that a sufficiently high mesh resolution in the shear layer is needed to correctly predict the dynamics of the shear layer. A too coarse resolution results in a stable shear layer and consequently in too low flow velocities and TKE in the cavity. A higher mesh resolution resulted in an improvement of the prediction flow field, but did not influence the dynamics of the shear layer further.

The capability of the FilteredNoise turbulent inflow generator was evaluated by comparing the cavity flow simulation with PIV measurements, featuring a turbulent inflow boundary layer. The FilteredNoise inflow generated showed capable of providing instationary turbulent inflow conditions that resulted in a generally good agreement of the simulated cavity flow field with the measured flow field. Negative effects due by the loss of TKE that was observed in the study of the artificial inflow turbu-
lence in Chapter 6 were not observed for the cavity flows. Consequently, the simulations could be used to perform a flushing study, revealing a mechanism that leads to two distinct phases with different removal rates. Additionally, the validation showed the need for high quality, time resolved stereo-PIV data. This allowed to perform a good LES by choosing the right mesh resolution. The high resolution PIV data enabled to specify the correct inflow conditions: for the laminar case it was important to have near wall velocity data (available due to the laser reflection mitigation in the wind tunnel measurements) and for the turbulent case, the time-resolved and stereo data made it possible to provided data to the turbulent inflow generator.

Finally, the urban air model was applied to a full scale case, an apartment building in an urban context. The use of an inflow generator provided the required turbulent ABL inflow conditions, leading to a short flow field initialization time. Additional surrounding buildings provided modifications to the otherwise streamwise aligned ABL mean flow field. In addition, the upstream building acted as a turbulence generator, shedding coherent structures that interact with the downstream building. This demonstration case showed that a domain with a limited spatial extent creates problems for the inflow generator. The streamwise fluctuations provided by the inflow generator were modified by the mass flow correction. This problem can be attributed to the large scale structures generated in the upper part of the ABL that are in the order of the domain size. The domain should be extended laterally and vertically to a multiple of the largest integral length scale. This can be done with a coarser grid and does not significantly increasing the cell count.
**DISCUSSION AND CONCLUSION**

This chapter contains a summary of the findings of the experimental and numerical studies that were conducted in this thesis. The contributions to the research field are stated, followed by a discussion of future work that could lead to better research tools and methods for the urban flow field analysis and ultimately a better understanding of the urban microclimate.

### 8.1 Summary

The aim of this thesis was to improve the understanding of urban flows on a local scale. The focus of the study was on the shear layer at the top of a street canyon in the skimming flow regime and under perpendicular flow conditions.

To this extent, wind tunnel tests at the ETHZ/EMPA Atmospheric Boundary Layer (ABL) wind tunnel were conducted. An experimental setup was developed that consists of a unit aspect ratio cavity, mounted flush on a split floor. Combinations of barriers and spires were used to generate different turbulent conditions upstream of the cavity. Without turbulence generators, the inflow is laminar. The flow field generated by the spires was measured using a Hot Wire (HW) probe. Time average and time resolved measurements of the cavity flow were performed using a stereoscopic Particle Image Velocimetry (PIV) system.

The average flow velocity of the cavity vortex was investigated for a large range of turbulent upstream conditions. It was found that a higher degree of turbulence upstream of the cavity resulted in a faster cavity vortex through a better transfer of momentum from the flow aloft to the cavity. The analysis of the turbulence production in the shear layer revealed three distinct spatial shapes. The majority of the moderately turbulent cases feature a thin elongated ellipse, located in the first half of the shear layer. Production in the second half can only be observed for the laminar inflow case. The cases with a high degree of turbulence show only short area of turbulence production at the beginning of the
shear layer. A detailed analysis of the time dependent behaviour for three cases, each representing one type of shear layer, was conducted. It was found that coherent structures at the cavity top are produced in each case, however the production location and mechanism is significantly different. The laminar case shows vortex shedding at a dominant frequency (self-sustained oscillations), the moderately turbulent cases feature vortices produced through Kelvin-Helmholtz instabilities and the case with high turbulence showed vortex shedding at the upstream edge, caused by sweep events. The time dependent analysis of the highly turbulent case showed intermittent, large scale sweep events that penetrate into the cavity. Additional vortices are formed at the interface between these high velocity areas and the low velocity cavity flow. The case with a large degree of turbulence is believed to be comparable to a street canyon located in the wake flow of an upstream building.

The cavity flow was further studied using Large Eddy Simulation (LES). The LES model uses the Finite Volume Method (FVM) approach and a one-equation subgrid scale model. Second order discretization in time and space is used. A validation of the LES model was performed by a comparison to wind tunnel experiments. Using an appropriate grid resolution, the model is capable of simulating the dynamics of the self-sustained oscillations observed in the experiment. A turbulent inflow generator based on the filtered noise method was used to simulate a moderately and highly turbulent cavity flow case. A good agreement with the wind tunnel data was achieved, showing the capability of the turbulent inflow generation. The ventilation of the street canyon was investigated by the flushing of a passive scalar. The investigated cases showed two distinct flushing rates. A first, faster removal rate was attributed to the removal of scalar through the shear layer. A second, slower removal rate was attributed to the transport of scalar from the center of the cavity vortex towards the outside, with subsequent removal through the shear layer. The difference between the two removal rates was the smallest for the highly turbulent case, due to the larger mixing of the external turbulence with the cavity vortex.

Furthermore, a demonstration case for an apartment building showed the potential of LES in combination with a turbulent inflow generator to study local scale turbulent flow phenomena in urban situations. The importance of including additional surrounding buildings was demon-
strated. Surrounding buildings influence the local mean flow field and generate additional coherent turbulent structures.

### 8.2 Contributions to the Research Field

Wind tunnel studies on street canyons are commonly conducted using a simulated ABL flow and an isolated street canyon, or a turbulent flow developed by a streamwise repetition of street canyons. This thesis presented a different approach to the study of the street canyon flow field. The wide range of turbulent flow conditions created by the wind tunnel setup revealed different types of shear layers. The experiment showed that large scale turbulence structures have a significant effect on the shear layer. The time resolved measurements showed large intermittent events and revealed the shedding of vortices at the upstream edge associated with sweep events. Statistical analysis tools were not able to show such intermittent events. Spatio-temporal visualizations of vortex cores using the Q criterion superimposed on sweeps and ejections, provided useful insights in the dynamics of shear layers and the interaction with external turbulence.

Through the validation with wind tunnel measurements, the use of high quality LES together with artificial turbulence generators, was found to be a viable approach to study the urban flow field. This was demonstrated through the use of passive scalar transport to investigate the ventilation of a street canyon and by the simulation of the urban flow field around an apartment building.

The use of time resolved techniques prompted the development and improvement of methods:

- Method to estimate of the uncertainty of integral length- and time scales obtained from time resolved PIV data using time-series analysis
- Efficient OpenFOAM implementation of the filtered noise turbulent inflow generator, including generation of turbulent scalar fields
- Robust method to determine the needed initialization time for LES based on passive scalar transport
Discussion and Conclusion

- Recommendations for the domain size for urban cases with LES and turbulent inflow generation

Additional contributions to the research community were made by developing open source tools to analyze turbulent flow data. These tools can import data from simulations performed with OpenFOAM, PIV measurements and HW measurements and store them in a common, non-proprietary file format. The analysis of the raw data can then be conducted using the provided analysis methods. This facilitates the direct comparison between data from measurements and simulations.

8.3 Outlook and Future Work

Large scale aspects of the urban microclimate such as the urban heat island effect or air pollution, are caused at the local scale. Solar radiation heats up individual building facades, roads and other parts of the urban landscape. Pollutants are generated by traffic at the street level or by building stacks on rooftops. Heat and pollutants are transported by the air through turbulent structures to other areas and to the ABL flow aloft.

A better understanding of the local flow field will help to improve the modelling and parameterization of local effects for large scale simulations that are unable to resolve the physical effects at the local scale. Furthermore, studies conducted at the local scale can identify areas where heat and pollutants might get trapped and investigate mitigation measures. The measurements and simulations presented in this thesis have shown that ventilation is governed by turbulent coherent motions. These can only be studied by methods that actually resolve the governing scales of turbulence, such as LES.

This section proposes some areas of research that could lead to an improvement of methods useful to study local scale turbulent flows.

The street canyon flow field investigated in this thesis represents a very specific case. The assumption that an urban morphology can be simplified to a street canyon with flat roofs of equal heights is questionable. Further work is required that shows to what extent the findings made by investigating street canyons are transferable to real urban cases. Further experimental investigations using time resolved imaging techniques such as PIV, or with high quality LES simulations could be conducted on specific types of urban morphologies to show if shear layers spanning from
rooftop to rooftop can exists and if they are the dominant flow feature that determines the exchange of air, heat and humidity between the street level and the flow aloft. Experimental investigations should not just focus on the flow field, but include a transport quantity that relates the flow field (cause) with the removal/entrainment of a scalar (effect). Instead of using point probes, i.e Fast Response Flame Ionization Detector (FFID), to measure the scalar concentrations, scalar transport can be investigated similarly to PIV on two-dimensional planes, using imaging techniques such as Quantitative Light Sheet (QLS) or Laser Induced Fluorescence (LIF) (for water tunnel facilities). Additionally, PIV seeding particles themselves could be used as a passive tracer. PIV image recording conducted during the rinsing of the wind tunnel with fresh air showed that PIV particles remained trapped inside the cavity before being completely ventilated into the wind tunnel. This shows that PIV particles could be used to study ventilation.

Dynamic analysis techniques that rely on time resolved field data, such as Dynamic Mode Decomposition (DMD), could provide further insight into the dynamics of urban flows. DMD can be successfully applied to flows that exhibit a dominant flow feature such as periodic vortex shedding. For fully turbulent flows with intermittent events, the interpretation of DMD results is difficult and further research is needed.

Even though LES in combination with an artificial turbulence generator proved successful in simulating the cavity flow experiment, further research on the method is required. If LES is to be applied to study more aspects of the urban microclimate, such as the transfer of heat from the buildings to the flow and vice versa, the wall boundary layer needs to be accurately captured. For full scale simulations of urban situations, the Reynolds number is high and the required wall resolution approaches Direct Numerical Simulation (DNS) requirements. This is, at least for the present time and near future due to limitations in computational resources, not possible. An option is to use wall modelling or a hybrid LES / Reynolds-Averaged Navier-Stokes (RANS) approach. This is an active field of research that could benefit from high quality experimental validation data for urban flows. Another approach to the wall problem could be to conduct urban LES simulations, similarly to wind tunnel tests, on a reduced scale, i.e. a reduced Reynolds number. Due to the established practice of scale model wind tunnel testing, Reynolds number effects have been researched for various cases. The Richardson number similarity
could still be fulfilled, as LES is not restricted to certain temperature limits in contrast to wind tunnel experiments.

Further research in the area of turbulent inflow boundary conditions, an important aspect of urban LES, is needed. Artificial inflow generators are capable of providing turbulent conditions that closely match desired conditions, are efficient and widely applicable. The filtered noise method showed to be a good candidate for a turbulent inflow condition, able to reproduce both conditions found in wind tunnel tests and ABL simulations. Further research is needed on the problem of loss in magnitude of imposed vertical and lateral fluctuations, occurring in the first cell after the inlet and whether or not this is connected to the divergence of the imposed velocity field. In addition, the turbulent generator needs a lot of information on the flow to be simulated. In addition to flow statistics (profiles of mean velocity and profiles of the full Reynolds stress tensor) also profiles of integral length scales (or correlation functions) for all three velocity components in all three directions (a total of nine profiles) are required. The usability of the inflow generator method for urban flows could be improved by creating and validating complete datasets for various ABL and Urban Boundary Layer (UBL) flows. The datasets can be created in a way to correct for the loss of Turbulent Kinetic Energy (TKE). The required length of the development fetch and the minimum grid resolution for each boundary layer case could be researched and provided as recommendations together with the dataset. In addition to input data, the method could be further improved by investigating the influence of choosing a higher order model for the streamwise correlation on the development of the artificial turbulence and the impact on the computational efficiency.

Further research is needed to develop means of quality control for LES. While the statistical accuracy was addressed in this thesis by providing a method to determine the minimum required initialization time, a major topic in LES that can benefit from further research is the minimum required grid resolution. A combination of simulation studies supported by experimental data could lead to guidelines appropriate for specific problems in urban microclimate investigations. Providing ABL and UBL datasets for the turbulent inflow generation would provide specific situations for which recommendations could be made.
Bibliography


open cavity with a laminar and a fully turbulent incoming boundary layer.” In: Journal of Fluid Mechanics 561, p. 113. ISSN: 0022-1120. DOI: 10.1017/S0022112006000735.


Cheng, Y. et al. (2003). „A comparison of large Eddy simulations with a standard k-e Reynolds-averaged Navier-Stokes model for the prediction of a fully developed turbulent flow over a matrix of cubes.” In: Journal of Wind Engineering and Industrial Aerodynamics 91.11, pp. 1301–1328. ISSN: 0167-6105. DOI: 10.1016/j.jweia.2003.08.001.


Bibliography


Richardson, L. (1922). Weather prediction by numerical process. Cambridge University Press.


