A Lattice-Boltzmann and Particle Image Velocimetry Study of Bounded Oscillatory Flow

Steven James Tonge

A thesis submitted in fulfilment of the requirements for the degree of Doctor of Philosophy to the University of Edinburgh

2005
Abstract

Bounded oscillatory flows occur in many situations. The lattice-Boltzmann model (LBM) is used to simulate oscillatory flows into the intermittently turbulent regime, in both two- and three-dimensions. The result of one of the three-dimensional simulations is also compared to Particle Image Velocimetry (PIV) measurements of an equivalent flow.

After a review of some theory of oscillatory flow and also the methods used in the LBM simulations and PIV measurements, a two-dimensional LBM is used to simulate oscillatory flow in an infinite two-dimensional channel. The development of turbulence over the oscillatory cycle is observed and presented. A three-dimensional LBM simulation of oscillatory flow in an infinite three-dimensional channel is performed and the results presented and compared to results from the infinite two-dimensional channel. A further simulation is performed with the three-dimensional LBM applied to an infinite square duct. The development of turbulence is again observed and compared to the LBM simulation in an infinite three-dimensional channel. The results of PIV measurements of a flow equivalent to that in the LBM simulation in an infinite square duct are then presented and compared to the results of the three-dimensional infinite square duct simulation. The results of this work clearly show that the LBM is a useful tool for simulating bounded oscillatory flows and also provides a clear insight into the transition to turbulence in this type of flow.
Declaration

I do hereby declare that this thesis was composed by myself and that the work described within is my own, except where explicitly stated otherwise.
Acknowledgements

I would first like to thank my supervisors Clive Greated and Murray Campbell for their help, enthusiasm and patience throughout this project. I am also grateful to other members of the Fluid Dynamics and Acoustics group over the course of this PhD, most notably Jim Buick, John Cosgrove and Paul Stansell for their input and advice. Also thanks to the rest of the fluids crew: Alan, Alan, Alistair, Callum, Darren, Mike, Orlando, Rob, Sandra, Seona, Ted and of course Dave. Further work-based thanks to all who have provided technical and moral support throughout this traumatic time, I won’t list you all in case I forget someone, but you know who you are.

More social thanks go to all members of the $E = MCC$ cricket team and also to all members of the rockin’ Physics social scene who were there at some of the hardest times when it felt like the end was never going to come. Thanks also to Bob, Gail, Jane, Jennie, Katharine, Ruth, Steven and Tom who have shared my homelife and coped with my various bouts of enthusiasm and despair. A huge thanks to my various internet buddies who have kept me entertained when all seemed lost in the middle of the day and provided many long distance laughs.

Finally thanks must go to my brother and sister, Paul and Clare, and my long suffering parents who have funded, cheered and grumbled their way through this entire enterprise. I can only hope they find the graduation ceremony worth it.
This thesis is dedicated to

The Hoff
Contents

1 Introduction ..................................... 1

1.1 Aims and Content ............................. 3

2 Oscillatory Flow Theory ....................... 7

2.1 Oscillatory Channel and Duct Flow Theory: Parameters and Regimes 7

2.2 The Laminar Regimes .......................... 10

2.2.1 Analytical Solution of the Navier-Stokes Equation for Laminar Oscillatory Flow in Two-Dimensions .......................... 10

2.2.2 Analytical Solution of the Navier-Stokes Equation for Laminar Oscillatory Flow in Three-Dimensions .......................... 12

2.3 The Turbulent Regimes ....................... 13

2.3.1 Averages in Turbulent Statistics ............. 14
2.3.2 Basic Turbulent Statistics ........................................ 14
2.3.3 Turbulent Correlations ............................................. 17
2.3.4 Turbulent Spectra and the Energy Cascade .................... 18
2.3.5 Length Scales ..................................................... 23

3 The Lattice Boltzmann Model ........................................... 25

3.1 History of the Lattice Boltzmann and the Lattice Gas Cellular Automata .................................................. 26

3.2 Lattice Dynamics ..................................................... 26

3.2.1 Two Dimensions: The D2Q9 Lattice ............................ 28

3.2.2 Three Dimensional Lattices ...................................... 29

3.3 The Lattice Boltzmann Equation .................................... 34

3.3.1 The Collision Function .......................................... 35

3.3.2 Streaming ......................................................... 37

3.4 Boundary Conditions .................................................. 38

3.4.1 Halfway Bounce Back Boundary .................................. 38

3.5 Driving the Oscillation .............................................. 40
3.6 Initialising the Laminar Regime ........................................... 42
3.7 Triggering the Turbulent Regime ........................................... 42
3.8 Flow Domains ..................................................................... 44
3.9 Parallel Implementation of the Model .................................... 45
   3.9.1 Parallel Computer Choices ........................................... 48

4 Particle Image Velocimetry Method ........................................ 55
   4.1 PIV Background ............................................................... 56
   4.2 PIV Technique ................................................................. 56
   4.3 Image Acquisition ............................................................. 57
      4.3.1 Flow Illumination ....................................................... 57
      4.3.2 Camera ...................................................................... 59
      4.3.3 Fluid ........................................................................ 59
      4.3.4 Seeding Particles ....................................................... 61
      4.3.5 Driving the Flow ....................................................... 61
      4.3.6 Timings ..................................................................... 61
      4.3.7 Image Capture ............................................................. 63
4.3.8 Experimental Setup ........................................ 64

4.4 Image Analysis .............................................. 65

4.4.1 Cross-correlation ....................................... 65

4.4.2 Peak Detection ........................................... 67

4.4.3 Image Analysis Errors .................................. 68

4.4.4 Improving Correlation .................................... 71

5 Presentation of Two-Dimensional Simulation Results 75

5.0.5 Simulation Parameters ................................... 76

5.1 Comparison of Simulation Results with Analytical Solution of the Navier-Stokes Equation .............................. 77

5.2 Development of Turbulence in Two-Dimensional Lattice-Boltzmann simulations .................................................. 80

5.2.1 Reynolds Stress and Turbulent Energy Cascade ........... 97

5.3 Conclusions .................................................. 101

6 Presentation of Three-Dimensional Channel Simulation Results 103

6.1 A Three-Dimensional Channel Model .......................... 103
6.1.1 Simulation Parameters ........................................ 104

6.2 Development of Turbulence in Three-Dimensional Channel Lattice-Boltzmann simulations ........................................ 105

6.2.1 Reynolds Stress and Turbulent Energy Cascade ............... 118

6.3 Conclusions .................................................................. 122

7 Presentation of Three-Dimensional Duct Simulation Results 123

7.1 A Three-Dimensional Duct Model .................................. 123

7.1.1 Simulation Parameters ............................................ 125

7.2 Development of Turbulence in three-dimensional duct Lattice-Boltzmann simulations ................................................. 125

7.2.1 Reynolds Stress and Turbulent Energy Cascade ............... 138

7.3 Conclusions .................................................................. 141

8 Presentation of Particle Image Velocimetry Results 143

8.1 Location of PIV Measurements ...................................... 144

8.2 Comparison of PIV measurement and Lattice-Boltzmann of a Turbulent Burst ................................................................. 146

8.2.1 Reynolds Stress and Turbulent Energy Cascade ............... 153

xiii
8.3 Conclusions ........................................ 157

9 Conclusions and Future Work 159

9.1 Conclusions ........................................ 159

9.2 Future Work ........................................ 160

9.2.1 Extension of Computational Simulations .......... 160

9.2.2 Extension of PIV Measurements ...................... 162

10 Publications 165

10.1 Journal Papers ...................................... 165

10.2 Book Chapters ...................................... 166

10.3 Conference Publications ............................. 166
List of Figures

2.1 Diagram of the computational domain for the three-dimensional channel simulations, three-dimensional simulations and PIV measurements. ................................. 20

3.1 Layout of the discrete velocity vectors on the D2Q9 lattice configuration. .......................................................... 28

3.2 Layout of the discrete velocity vectors on the D3Q27 lattice configuration. .......................................................... 30

3.3 Layout of the discrete velocity vectors on the D3Q19 lattice configuration. .......................................................... 32

3.4 Layout of the discrete velocity vectors on the D3Q15 lattice configuration. .......................................................... 33
3.5 Graphical representation of two sample particle distributions, coloured red and blue, streaming to the boundary and returning with the halfway bounce back boundary condition. (a) shows the position of the two distributions at timestep $t$. (b) shows the positions at timestep $t+1$ as they approach the boundary. (c) shows them after bouncing off the boundary back in the direction they came from. ............................................................... 39

3.6 Diagram of the computational domain for the two-dimensional simulations. ......................................................... 44

3.7 Diagram of the computational domain for the three-dimensional channel simulations, three-dimensional simulations and PIV measurements. ................................................................. 46

3.8 Diagram of the three-dimensional computational domain, indicating the processor boundaries for a four processor split in the streamwise direction. ................................................................. 52

3.9 Diagram of the three-dimensional computational domain, indicating the processor boundaries for an eight processor split. ........ 53

4.1 Timeline for PIV experiment. Note that timeline is not to scale. . 62

4.2 Schematic diagram of the Image Acquisition setup. ............... 64

xvi
5.1 Comparison of streamwise velocity, $u$, profile in a lattice-Boltzmann simulation and the analytical solution to the Navier-Stokes equation at phases $45^\circ(a)$, $90^\circ(b)$, $135^\circ(c)$ and $180^\circ(d)$ with $\alpha = 7$ and $Re_\delta \sim 710$ .......................................................... 78

5.2 Comparison of streamwise velocity, $u$, profile in a lattice-Boltzmann simulation and the analytical solution to the Navier-Stokes equation at phase $180^\circ$ with $\alpha = 7$ and $Re_\delta \sim 710$ showing the absolute difference. .......................................................... 79

5.3 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer. .......................................................... 82

5.4 Plot showing the mean streamwise velocity of a laminar flow and the scaled forcing term with $\alpha = 7.0$, $Re_\delta = 918.6$. The forcing term has been scaled to the mean streamwise velocity for comparison, but it actually varies in the range $-P_{GRAD} \leq F \leq P_{GRAD}$ .... 83
5.5 Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_\delta = 918.6$ and $\omega t = 2.356(a)$, 2.749(b), 2.945(c), 3.142(d), 3.587(e), 3.927(f), 4.320(g) and 4.712(h). The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 5.3. Therefore, the normalised vector magnitude, is calculated as

$$|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2}}{|u|}.$$

5.6 Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_\delta = 918.6$ and $\omega t = 5.498(a)$, 5.890(b), 6.087(c), 6.283(d), 6.676(e), 7.069(f), 7.461(g) and 7.854(h). The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 5.3. Therefore, the normalised vector magnitude, is calculated as

$$|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2}}{|u|}.$$

5.7 Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_\delta = 918.6$ and $\omega t = 8.639(a)$, 9.032(b), 9.228(c), 9.425(d), 9.817(e), 10.210(f), 10.603(g) and 10.996(h). The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 5.3. Therefore, the normalised vector magnitude, is calculated as

$$|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2}}{|u|}.$$
5.8 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at ±0.80 indicate the width of the Stokes boundary layer. .......................................................... 94

5.9 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at ±0.80 indicate the width of the Stokes boundary layer. .......................................................... 95

5.10 Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at ±0.80 indicate the width of the Stokes boundary layer. .......................................................... 99

5.11 A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 7.0$, $Re_\delta = 918.6$. .............. 100

6.1 Diagram of the computational domain for the three-dimensional channel simulations. .......................................................... 104

6.2 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 987.47$. The horizontal lines at ±0.80 indicate the width of the Stokes boundary layer. .......................................................... 107
6.3 Contour plot of the normalised turbulent intensity, $q^*$ across the
width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_5 = 918.6$. The
horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary
layer. ................................................................. 108

6.4 Contour plot of the normalised components of the velocity, $u^*(a)$,
v$^*(b)$ and $w^*(c)$ across the width of the channel width $\omega t$ with
$\alpha = 7.0$, $Re_5 = 987.47$. The horizontal lines at $\pm 0.80$ indicate the
width of the Stokes boundary layer. .............................. 111

6.5 Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_5 = 987.47$
and $\omega t = 2.553(a)$, 3.142(b), 3.731(c), 4.320(d) and 4.909(e). The
normalised magnitude of the vectors, $|u'|^*$, is indicated by their
colour, which is scaled so as to be comparable to $q^*$ in figure
6.2. Therefore, the normalised vector magnitude, is calculated
as $|u'|^* = \frac{|u'|}{|u|} = \sqrt{v'^2 + w'^2 + u'^2}$. The size of the vectors is scaled by
their relative magnitude. ............................................. 117

6.6 Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across
the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_5 = 987.47$. The
horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary
layer. ................................................................. 120
6.7 A plot of the turbulent energy spectra, $E(k)$ at different values of
$\omega t$, indicated in the legend, with $\alpha = 7.0, Re_\delta = 987.47$.

7.1 Diagram of the computational domain for the three-dimensional
duct simulations.

7.2 Contour plot of the normalised turbulent intensity, $q^*$ across the
width of the channel width $\omega t$ with $\alpha = 7.0, Re_\delta = 884.77$. The
horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary
layer.

7.3 Contour plot of the normalised components of the velocity, $u'^*(a)$,
$v'^*(b)$ and $w'^*(c)$ across the width of the channel width $\omega t$ with
$\alpha = 7.0, Re_\delta = 884.77$. The horizontal lines at $\pm 0.80$ indicate the
width of the Stokes boundary layer.

7.4 Vector maps of the fluctuating velocities with $\alpha = 7, Re_\delta = 884.77$
and $\omega t = 2.945(a), 3.534(b), 4.123(c), 4.712(d)$ and $5.301(e)$. The
normalised magnitude of the vectors, $|u'|^*$, is indicated by their
colour, which is scaled so as to be comparable to $q^*$ in figure
6.2. Therefore, the normalised vector magnitude, is calculated
as $|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2 + w'^2}}{|u|}$. The size of the vectors is scaled by
their relative magnitude.
7.5 Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm0.80$ indicate the width of the Stokes boundary layer. .......................................................... 139

7.6 A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 7.0$, $Re_\delta = 884.77$. ................. 140

8.1 Diagram showing example PIV slices taken through the experimental duct. .......................................................... 145

8.2 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 6.76$, $Re_\delta = 824$. The horizontal lines at $\pm0.80$ indicate the width of the Stokes boundary layer. .......................................................... 147

8.3 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 824$. The horizontal lines at $\pm0.80$ indicate the width of the Stokes boundary layer. .......................................................... 148

8.4 Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 6.76$, $Re_\delta = 824$. The Stokes boundary layer is not visible, due to its small width in the experimental situation. .......................................................... 152
8.5 Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 6.76$, $Re_\delta = 824$. The horizontal lines at ±0.80 indicate the width of the Stokes boundary layer. ........................................ 155

8.6 A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 6.76$, $Re_\delta = 824$. ............ 156
List of Tables

2.1 Ranges of $Re_\delta$ for the different flow regimes in oscillatory flow... 9

4.1 Full setting for BNC 500 signal generator. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 63

5.1 Parameters for two-dimensional simulations. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 76

6.1 Parameters for three-dimensional channel simulation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 105

7.1 Parameters for three-dimensional duct simulation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 125
Chapter 1. Introduction

Introduction

Oscillatory flows occur all around us in both natural and man-made circumstances, yet despite this, there have been relatively few studies made of them. They appear in various situations on many different length scales. Important examples include tidal and oceanic flows, arterial flows and all acoustical flows, as well as in many industrial pumping and mixing applications. In almost all of these situations, the flow is bounded on at least one side. An important feature of oscillatory flows is the transition to turbulence. However very little is known about the details of this transition. Theoretical studies on the transition from a laminar flow profile to a turbulent one in oscillatory flows have been fairly limited due to the mathematical complexities of the situation, with most studies only considering two-dimensional cases.

While it is possible to accurately measure the flow field of a real fluid us-
ing a non-invasive technique, computer simulations can potentially give an easily reproducible and much more detailed picture of what is occurring. Several computational methods have been proposed over the years with varying degrees of success and accuracy. These simulation techniques can be broadly divided into two groups, direct numerical and Fourier methods. Direct numerical simulations attempt to provide a real space numerical solution to the Navier-Stokes equations for a given flow. Simulations using spectral methods also attempt to solve the Navier-Stokes equation, but in inverse or frequency space. Both of these approaches to the problem make quite large simplifications in the mathematics of the flow in order to arrive at an approximate solution in a reasonable time frame, especially for flows in the turbulent regime. Indeed, the complex nature of turbulent flows mean that it has only recently become possible to simulate a turbulent flow over multiple length scales. Most previous studies have been spectral methods looking at only a few specific frequencies, the most prominent example for bounded oscillatory flows being by Akhavan et al[1].

The lattice-Boltzmann method has one main advantage over other computational techniques. It gives a solution to the Navier-Stokes equation to second order accuracy from a first-order computation[2]. This gives the lattice-Boltzmann model an efficiency advantage over other numerical methods, allowing much complete modelling of the fluid for an equivalent amount of computation compared to other methods.
Chapter 1. Introduction

1.1 Aims and Content

The main objective of the work described in this thesis is the development of a three-dimensional lattice-Boltzmann model suitable for simulating bounded oscillatory flows into the turbulent regime. This overall aim can be broken down into four main steps:

i) Show that a two-dimensional lattice-Boltzmann simulation can successfully model the turbulent regime in a bounded oscillatory flow.

ii) Develop a full three-dimensional lattice-Boltzmann model.

iii) Show that a three-dimensional lattice-Boltzmann simulation can successfully model the turbulent regime in a bounded oscillatory flow.

iv) Compare the results of the lattice-Boltzmann simulation to laboratory measurements of an equivalent flow.

This thesis is split into two parts. Chapters 2, 3 and 4 of the thesis cover the theory and techniques used in the simulations, measurements and analysis, while the results of the simulations and measurements are presented in chapters 5, 6, 7 and 8.

Chapter 2 covers the basic theory of oscillatory flow, both in the laminar and turbulent regimes. It also includes information on how the turbulence statistics for the simulations and measurements were calculated.
Chapter 3 describes the lattice-Boltzmann method. It includes an explanation of the model and information on how the flow was driven and bounded. It also includes information on the computing facilities used and the setup and configuration of the lattice.

Chapter 4 contains information on the setup and techniques used in the PIV measurements, including a brief history of PIV and information on the equipment used in the measurements.

Chapter 5 presents the results of the two-dimensional simulations. These simulations show that it is possible to simulate a bounded oscillatory flow in the turbulent regime with the lattice-Boltzmann. The reduced runtime of a two-dimensional simulation compared to a three-dimensional one also allows for a look at the development of the flow over several oscillations, rather than just the one or two possible in three dimensions.

Chapter 6 presents the results of the three-dimensional channel simulation. This is a step between the two-dimensional simulations and the three-dimensional duct simulation. The two-dimensional simulations and the three-dimensional channel simulation share the same laminar flow profile, and this allows for a direct comparison as turbulence develops.

Chapter 7 presents the results of the three-dimensional duct simulation. The results of a full three-dimensional bounded oscillatory flow in a square duct are
presented in this chapter, for direct comparison with the PIV measurements.

Chapter 8 presents the results of the PIV measurements. The parameters of the flow measured here have been chosen so as to closely match those from the three-dimensional duct simulation for a direct comparison.

Chapter 9 then summarises the conclusions from each of the results chapters and provides some discussion of problems and shortcomings of the work and ideas and directions for future investigation.
Chapter 2

Oscillatory Flow Theory

This chapter provides some basic theory of oscillatory flows and explains how the turbulence statistics for the simulations and measurements were calculated.

2.1 Oscillatory Channel and Duct Flow Theory: Parameters and Regimes

For oscillatory flow in channels and rectangular ducts, there are two main non-dimensional parameters. These are the Womersley parameter, $\alpha$ and the Reynolds number based on the Stokes boundary layer thickness, $Re_\delta$. These
parameters are defined as follows:

\[ \alpha = r \sqrt{\frac{\omega}{\nu}} \]  

(2.1)

\[ Re_\delta = \frac{|u_0| \delta}{\nu} \]  

(2.2)

For the Womersley parameter, \( r \) is the half channel width, \( \omega \) is the frequency of the oscillation and \( \nu \) is the kinematic viscosity of the fluid. For the Reynolds number based on the Stokes boundary layer thickness, \( |u_0| \) is the maximum magnitude of the flow velocity over the whole oscillation and \( \delta \) is the Stokes boundary layer thickness. This is defined as:

\[ \delta = \sqrt{\frac{2\nu}{\omega}} \]  

(2.3)

One further parameter which is often used instead of \( \alpha \) is the Stokes parameter, \( \Lambda \). This is defined as:

\[ \Lambda = \frac{r}{\delta} = \frac{\alpha}{\sqrt{2}} \]  

(2.4)

The Womersley parameter is a non-dimensional parameter which governs the relationship between the unsteady and viscous forces. The Reynolds number is a measure of the stability in a flow and indicates when the transition from laminar to turbulent flows will occur. More generally, the Reynolds number, \( Re \), of a flow is defined as:

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

(2.5)
Chapter 2. Oscillatory Flow Theory

<table>
<thead>
<tr>
<th>$Re_\delta$ Range</th>
<th>Flow Regime</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_\delta &lt; 100$</td>
<td>Laminar</td>
</tr>
<tr>
<td>$100 &lt; Re_\delta &lt; 500$</td>
<td>Disturbed Laminar</td>
</tr>
<tr>
<td>$500 &lt; Re_\delta &lt; 2000$</td>
<td>Intermittently Turbulent</td>
</tr>
<tr>
<td>$Re_\delta &gt; 2000$</td>
<td>Turbulent</td>
</tr>
</tbody>
</table>

Table 2.1: Ranges of $Re_\delta$ for the different flow regimes in oscillatory flow.

Here, $U$ and $L$ are the characteristic velocity and length for a given flow. For oscillatory flow, these are taken as the maximum velocity taken over the whole oscillation and the Stokes boundary layer thickness. It is a widely accepted view that turbulence in a bounded oscillatory flow is governed by the Reynolds number based on the Stokes boundary layer width as long as the width of the Stokes boundary layer is small compared to the overall channel width[3, 4, 5]. Hence the Stokes boundary layer width is used instead of the half-channel width, $r$, as the characteristic length in the calculation of the Reynolds number. The exact value of $Re_\delta$ at which the transition to turbulence occurs is not precisely defined, and it can be greatly influenced by the experimental or computational setup used to measure it. It has been suggested that there are four separate flow regimes, shown in table 2.1 with the approximate range of $Re_\delta$ at which they occur[6].

In the laminar regime, the flow does not vary significantly from the analytical solutions for the Navier-Stokes equation. In the Disturbed Laminar regime, there are noticeable perturbations to the laminar flow described analytically, but little statistical evidence of turbulence in the flow. In the Intermittently Turbulent regime, bursts of turbulence appear at specific phases in the oscillatory cycle, with the flow re-laminarising. In the Turbulent regime, the flow is turbulent at
2.2 The Laminar Regimes

Laminar flow is a much simpler flow regime than a turbulent one. In certain cases this allows an exact solution to the Navier-Stokes equation to be calculated. Oscillatory flow in a channel or square duct are two of these cases.

2.2.1 Analytical Solution of the Navier-Stokes Equation for Laminar Oscillatory Flow in Two-Dimensions

Here we consider an analytical solution for flow in an infinitely long two-dimensional channel of width $2r$. The fluid is driven by a sinusoidally oscillating pressure gradient:

$$\frac{\partial p}{\partial x} = -P_{GRAD} e^{i\omega t}$$

(2.6)

Here, $P_{GRAD}$ is the amplitude of the pressure gradient. The $x$-axis is parallel to the channel walls, which is known as the streamwise direction. The $y$-axis is perpendicular to the walls and is known as the spanwise direction. The centre of the channel is defined as $y = 0$. It is clear that the resulting flow velocity will be independent of $x$ and that $u(y, t) = 0|_{y=\pm r}$. This gives a Navier-Stokes equation
in the form of:

\[
\frac{\partial u(y, t)}{\partial t} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \frac{\partial^2 u(y, t)}{\partial t^2}
\]  

(2.7)

where \( \rho \) is the fluid density and \( u(y, t) \) is the x-component of the velocity \( u \), with the y-component of the velocity, \( v = 0 \). Inserting the pressure gradient from equation 2.6 and looking for a solution of the form:

\[
u(y, t) = U(y) e^{i\omega t}
\]  

(2.8)

this reduces equation 2.7 to:

\[
\frac{\partial^2 U(y)}{\partial t^2} - \frac{i\omega t}{\nu} = -\frac{P_{\text{GRAD}}}{\rho \nu}
\]  

(2.9)

The final expression for the velocity is[7]:

\[
u(y, t) = \Re \left\{ \frac{P_{\text{GRAD}}}{i\omega \rho} \left[ 1 - \frac{\cosh\left(\frac{1}{\sqrt{2}}(\alpha + i\alpha)i\right)}{\cosh\left(\frac{1}{\sqrt{2}}(\alpha + i\alpha)\right)} \right] e^{i\omega t} \right\}
\]  

(2.10)

Where \( \Re \) denotes the real part of the solution.
2.2.2 Analytical Solution of the Navier-Stokes Equation for Laminar Oscillatory Flow in Three-Dimensions

In three-dimensions, the flow is in an infinitely long square duct of width $2r$, driven by the forcing term given in equation 2.6. The $x$-axis is parallel to the direction of flow, with the $y$- and $z$-axis being perpendicular to the streamwise direction. The solution to the Navier-Stokes equation was derived by Drake[8]. The solution is given below in the notation used here:

$$u(y, z, t) = \Re \{f(y, z)e^{i\omega t}\} \quad (2.11)$$

where:

$$f(y, z) = \frac{4PG_{R A D}r^2}{\pi \nu} \sum_{n=0}^{\infty} \frac{(-1)^n}{(2n+1)w^2} \left(1 - \frac{\cosh\frac{2\pi y}{r}}{\cosh w}\right) \cos \frac{2n+1}{2r} \pi y \quad (2.12)$$

with:

$$w = \left(\frac{i\omega r^2}{\nu} + \frac{(2n+1)^2}{4\pi^2}\right)^{\frac{1}{2}} \quad (2.13)$$

For calculations of the velocity profile used here, the series solution has been truncated at $n = 100$. A further solution has been derived by Fan and Chao[9], however Drake's solution was preferred, due to its more efficient computation and the difference between two solutions is negligible.
2.3 The Turbulent Regimes

Turbulence is a part of practically every fluids system that exists in everyday life, from blood flowing through the veins in our bodies to the movements of gases in the photosphere of the sun and interstellar clouds. Turbulence is used as an aid to mixing and is an integral part of the combustion process in most engines. While it is such a wide spread and important phenomena there is no universal mathematical description and due to its extremely complex nature, it seems very unlikely that one will be developed. Turbulence is very difficult to describe and study, both experimentally and also using computer simulations. In addition to experimental measurements, many different modelling techniques have been developed to try and understand the underlying mechanisms of turbulent flows. The main approaches to computer modelling of turbulence are direct numerical integration of the Navier-Stokes equations[10, 11], large-eddy simulations[12, 13], Reynolds-averaged Navier-Stokes[14] and the lattice-Boltzmann model[15], which is used in this study. There has also been some work done using Monte Carlo to simulate the underlying stochastic process[16, 17]. Most of the parameters used to describe turbulence involve some sort of statistical dependence and an explanation of the ones used in this investigation is provided below, with particular reference to the development of turbulence in oscillatory flows. As turbulent flows are highly unstable, a chaos analysis can also provide insights into the workings of turbulent flows[18, 19]
2.3.1 Averages in Turbulent Statistics

One of the main features of most turbulent statistics, particularly in time-independent flows, is averages which are taken over time. In this study, this is not possible, as oscillatory flows are time dependent. Although it would be possible to take temporal averages by gathering data over many oscillations and then averaging over points with the same phase in the oscillatory cycle, this cannot be achieved due to the large run-times involved in lattice-Boltzmann simulations of turbulent flows. Therefore, the vast majority of averages taken are spatial ones, so that the statistics calculated will not be distorted by the change in the mean flow velocity with time. These spatial averages are taken over a line of points in the streamwise direction of the flow, where the laminar velocities are all identical.

2.3.2 Basic Turbulent Statistics

Most information about turbulence is measured statistically, as the flow patterns of any given burst of turbulence are unique. It is therefore most useful to compare turbulent statistics of different situations. These statistics are normally calculated on the fluctuating part of the velocity, $u'$. $u'$ is calculated by removing the mean velocity from the total velocity. The fluctuating component of the velocity is calculated as follows:

$$ u'(r) = u(r) - \bar{u}_x(y, z) $$  \hspace{1cm} (2.14)
Chapter 2. Oscillatory Flow Theory

Note that in two-dimensions \( \bar{u}_x(y, z) \) only depends on \( y \) as there is no \( z \)-component in this case. The \( x \)-component of \( \bar{u}_x(y, z) \), \( \bar{u}_x(y) \), is calculated as follows:

\[
\bar{u}_x(y, z) = \frac{\sum_{x=1}^{N} u(r)}{N}
\]

(2.15)

where \( N \) is the number of data points in the \( x \)-direction. The calculation of the other two components of \( \bar{u}_x(y, z) \), \( \bar{u}_y(y, z) \) and \( \bar{u}_z(y, z) \) depends on the situation being studied and is detailed in the relevant sections. The main turbulent statistic used is the normalised turbulent intensity, \( q^* \), which is calculated as follows:

\[
q^*(y) = \frac{q}{|u|} = \sqrt{\bar{u}^2 + \bar{v}^2 + \bar{w}^2}
\]

(2.16)

Obviously, it is not necessary to include the \( z \)-component of the velocity in the two-dimensional cases. \( \bar{u}^2 \), \( \bar{v}^2 \) and \( \bar{w}^2 \) are the \( x \)-, \( y \)- and \( z \)-components of \( \bar{u}^2 \). As \( q^*(y) \) varies over the spanwise direction, the components of \( \bar{u}^2 \) will be calculated differently for the two-dimensional simulation than for the three-dimensional channel and three-dimensional duct ones. For the two-dimensional simulations:

\[
\bar{u}^2 = \frac{\sum_{x=1}^{N} u'^2(r)}{N}
\]

(2.17)

For the three-dimensional channel and three-dimensional simulations:

\[
\bar{u}^2 = \frac{\sum_{x=1}^{N} \sum_{z=1}^{L} u'^2(r)}{NL}
\]

(2.18)
where \( L \) is the number of data points in the \( z \)-direction. The normalisation constant, \( |\bar{u}| \), is the RMS velocity of the laminar profile taken over one oscillation for the given flow conditions. In the two-dimensional and three-dimensional channel simulations, this is defined as:

\[
|\bar{u}| = \sqrt{\int_0^{2\pi} \int_{-r}^r |u|^2 \, dy \, dt} \over 4\pi r \tag{2.19}
\]

For the three-dimensional channel and duct simulations, an extra integral is required to take account of the spatial variance in the \( z \)-direction:

\[
|\bar{u}| = \sqrt{\int_0^{2\pi} \int_{-r}^r \int_{-r}^r |u|^2 \, dy \, dz \, dt} \over 8\pi r^2 \tag{2.20}
\]

For the PIV measurements, it is impossible to measure the velocities of the laminar profile at the Reynolds numbers required for turbulent flow. The laminar profile used in this case was calculated from the analytical solution for the velocity profile given in section 2.2.2.

There are many further statistics that can be calculated, such as Skewness and Kurtosis statistics which give a further insight into the structures in the flow. However, these statistics are particular prone to errors arising from noise in PIV measurements and so would be no use for comparison with the simulation results[20].
2.3.3 Turbulent Correlations

Correlations give information on the velocity fluctuations and also the relation of the velocity components to each other at a given point in the flow. The prime example of this is the Reynolds stress, $\varphi$, which is defined as follows:

$$\varphi = -\rho u'v'$$  \hspace{1cm} (2.21)

Here, $\overline{u'v'}$ is $u'v'$ averaged over the streamwise direction:

$$\overline{u'v'} = \frac{\sum_{z=1}^{N} u'v'}{N}$$  \hspace{1cm} (2.22)

The value of $\varphi$ is intrinsically linked with turbulent shear flows, and is a key parameter in the understanding of turbulence, particularly in ducts with a non-circular cross section. When $u'v'$ is non-zero, this implies that the $u'$ and $v'$ components of $u'$ are not independent of each other. A change from a zero value to a non-zero value of $\varphi$ indicates a transition from laminar flow to turbulent flows as in laminar flow $v' \sim 0$ and therefore $\overline{u'v'} \sim 0$. The physical interpretation of a non-zero value of $\overline{u'v'}$ is as follows. When $\overline{u'v'} < 0$, at all instances where $u'$ is positive, $v'$ is more likely to be negative and vice-versa. A fluid particle moving with a positive $v'$ is likely to have a corresponding negative $u'$ indicating that it is arriving into its current position from an area of slower mean flow. This is also true of the correlation of the $u'$ and $w'$ components of $u'$ in flows in a square...
Chapter 2. Oscillatory Flow Theory

duct.

The normalised mean Reynolds stress, $\varphi^*$, is calculated differently, depending on whether it is being calculated for a two- or three-dimensional simulation, as is the case for $q^*$. For two-dimensional simulations, it is just the Reynolds stress normalised with $|\bar{u}|$:

$$
\varphi^* = \frac{\varphi}{|\bar{u}|} = \frac{-\bar{u}u'}{|\bar{u}|} \tag{2.23}
$$

For three-dimensional simulations, a further average is required due to the extra dimension:

$$
\varphi^* = \frac{\varphi}{|\bar{u}|} = \frac{\sum_{z=1}^{L} -\bar{u}u'}{|\bar{u}| L} \tag{2.24}
$$

The calculation of further correlations, such as a spatial correlation of two velocities, $u_iu_j$, can be used to further describe features of turbulence generated in a given flow. However, as this study is primarily concerned with the development of turbulence, the information provided by the Reynolds Stress is sufficient and the calculation of further correlations is unnecessary.

2.3.4 Turbulent Spectra and the Energy Cascade

Turbulence is widely characterised by the picture of vortexes forming in the flow. This is true to some extent and these three-dimensional localised structures do play an important part in the transfer of energy in turbulent flows. The energy cascade refers to the transfer of turbulent energy down the length scales from large
eddy which draw energy from the main flow through to the small scales where the energy is dissipated by the action of viscosity. The size of the largest eddies is usually comparable to the size of the system, though these are not necessarily the most energetic structures. The energy which eddies of different size contain is normally measured in reciprocal or $k$-space, with a schematic example of an energy spectra, $E(k)$, being shown in figure 2.1. There are three separate sections to the energy spectra of a turbulent flow, labelled in figure 2.1. The first is the Input Range. This contains the largest eddies (indicated by its placing in the smallest values of $k$) and is where energy fed into the flow enters the turbulent spectrum. These larger eddies generate smaller ones which transfers the energy down in a cascade. The next section is the Inertial Range, where all the incoming energy is provided by larger eddies and all the energy lost is passed to smaller eddies. The spectrum in the Inertial Range is associated with an exponent of $-\frac{5}{3}$, which was determined by Kolmogorov from dimensional analysis\[21, 22\]. He argued that the energy was only a function of the wave number and the global energy dissipation rate, $\epsilon$:

$$E(k) = C\epsilon^{\frac{3}{5}}k^{-\frac{5}{3}}$$  \hspace{1cm} (2.25)

$C$ and $\epsilon$ are constants for a given flow. $\epsilon$ is given by McComb\[23\]:

$$\epsilon = \int_0^\infty 2\nu k^2 E(k)dk$$  \hspace{1cm} (2.26)
Chapter 2. Oscillatory Flow Theory

Figure 2.1: Diagram of the computational domain for the three-dimensional channel simulations, three-dimensional simulations and PIV measurements.

\[ C = C_k \]

where \( C_k \), also known as \( C_1 \), is the Kolmogorov constant. It is thought that \( C_k \) has a universal value at asymptotically high Reynolds numbers and while some doubts have been raised over this universality, is still fairly well accepted[24]. \( C_k = 0.53 \pm 0.055 \) is the value given by Sreenivasan[25] which gives \( C = \frac{55}{18} C_k = 1.62 \). The final stage of the energy cascade is known as the Dissipation Range and here the energy is passed down from larger eddies in the inertial range and then dissipated by viscous forces at small scales.

Discrete Calculation of Turbulent Energy Spectra

As both the simulations and experiments produce discrete data, changes need to be made to the calculations used to compute the energy spectra. The Fourier
transform of the fluctuation velocities, $F(k)$, is calculated as follows:

$$F(k) = \sum_{x=0}^{N-1} u' e^{-i2\pi k x}$$  \hspace{1cm} (2.27)$$

Thus, $F(k)$ is a one-dimensional Fourier transform of the $x$-component of the fluctuation velocity, $u'$, in the streamwise direction. $F(k)$ is calculated for each location in the spanwise and, in three-dimensional cases, cross-stream directions. A mean of the transform at each location is then computed to give a single Fourier transform, $\overline{F}(k)$, of $u'$ for a given set of data. In two-dimensions:

$$\overline{F}(k) = \frac{\sum_{y=0}^{M-1} |F(k)|^2}{M}$$  \hspace{1cm} (2.28)$$

For the three-dimensional version of $\overline{F}(k)$, and additional average is required over the cross-stream direction:

$$\overline{F}(k) = \frac{\sum_{y=0}^{M-1} \sum_{z=0}^{L-1} |F(k)|^2}{ML}$$  \hspace{1cm} (2.29)$$

The value of the energy at each length scale is calculated as follows:

$$E(k) = 4\pi k^2 \overline{F}(k)$$  \hspace{1cm} (2.30)$$
Chapter 2. Oscillatory Flow Theory

The global energy dissipation rate, $\epsilon$, is calculated as follows:

$$\epsilon = 2\nu \sum_{k=0}^{K} k^2 E(k) \quad (2.31)$$

As the flows being studied here are time-dependent, the value of $\epsilon$ will change at different phases in the flow. As $E(k)$ is dependent on $\epsilon$ (equation 2.25), it is not possible to directly compare the energy spectra at different times. In order to do this, the value of $E(k)$ needs to be normalised for each time. By combining equations 2.25 and 2.30, it is possible to obtain the following relation:

$$\frac{E(k)}{C\epsilon^{\frac{2}{3}}} = \frac{4\pi k^2 \tilde{F}(k)}{C\epsilon^{\frac{2}{3}}} = k^{-\frac{5}{3}} \quad (2.32)$$

When plotting out the energy spectra, it is also useful to scale the values of $k$ with the Kolmogorov length, $l_k$, described in section 2.3.5. This gives the Kolmogorov length scaled inverse space, $k_d$:

$$k_d = kl_k \quad (2.33)$$

For convenience, the scaled values of $E(k)$ have been relabelled in the energy spectra plots so that:

$$E(k_d) = \frac{E(k)}{C\epsilon^{\frac{2}{3}}} \quad (2.34)$$

By plotting the values of $E(k)$ in this manner, it is possible to directly compare the energy spectra at different phases in the flow and also compare the cascade in the inertial sub-range against the Kolmogorov $-\frac{5}{3}$ law.
2.3.5 Length Scales

There are three important lengths when considering turbulence. These fall into two different length scales. The first contains two characteristic lengths of similar size. One is the integral length scale, \( l_0 \), and the other is the length at which the eddies contain the most energy, \( l_e \). The integral length scale is the largest length scale at which there is any correlation in the flow, and so is the size of the largest eddies. In turbulent channel and duct flow, \( l_0 \sim 2r[26] \). The largest eddies are not the most energetic and although \( l_e \) is of the same order of magnitude as \( l_0 \). In particular, it is normally found in the range \( \frac{r}{4} < l_e < \frac{r}{2} [26, 20] \). \( l_e \) occurs at the maximum value of \( E(k) \). The second and smaller length scale contains the final characteristic length and is known as the Kolmogorov length, \( l_k \). This length scale is associated with the smallest dissipative eddies at the bottom of the energy cascade and indicates where the viscous forces are very strong. It was derived by Kolmogorov from dimensional analysis and is given by:

\[
l_k = \left( \frac{\nu^3}{\epsilon} \right)^{\frac{1}{4}} \tag{2.35}\]

Kolmogorov also derived corresponding velocity and time scales:

\[
v_k = (\nu \epsilon)^{\frac{1}{4}} \tag{2.36}\]

\[
t_k = \left( \frac{\nu}{\epsilon} \right)^{\frac{1}{2}} \tag{2.37}\]
Chapter 3

The Lattice Boltzmann Model

The lattice Boltzmann model (LBM) is a relatively new technique for computational simulation of fluid dynamics. It evolved from the lattice gas cellular automata, and has several obvious advantages to that technique, which are outlined in section 3.1. Both of these models are derived from molecular dynamics techniques rather than computing the direct solution of the Navier-Stokes equation by some integration technique. It can be shown that the Navier-Stokes equation can be recovered from the lattice-Boltzmann\[27\]. The LBM has been used successfully to simulate a wide variety of fluid situations including acoustic and multiphase wave propagation\[28, 29\], magnetohydrodynamics\[30\], multiphase flow\[31\], and chemical reacting flows\[32\]. The LBM has also been used to simulate high Reynolds turbulent flows\[2, 33\], which it is applied to in this study.
3.1 History of the Lattice Boltzmann and the Lattice Gas Cellular Automata

The lattice gas cellular automata (LGCA) was already established as a useful computational technique for fluid dynamics when the application of lattice Boltzmann models were applied to fluid dynamics[15]. These had several advantages over the LGCA. The main one is that the LBM is Galilean invariant. The LBM also eliminated noise introduced by the statistical averaging and removed the look-up tables for the states of the lattice sites. The statistical noise was eliminated by using particle distributions at the lattice sites instead of Boolean variables. This also eliminated the need for averaging over several sites to produce the macroscopic variables. This greatly improved the models suitability for implementation on cheap, distributed memory parallel computers, such as Beowulf clusters, as it made that lattice interactions much more local. The number of states for the lattice sites was also reduced in the LBM. This is shown in section 3.3.1.

3.2 Lattice Dynamics

There are several different configurations in which the lattice and distribution velocities can be arrange in an LBM simulation. The most common ones in use at
the moment are based around a square lattice, although a hexagonal lattice has been favoured in the past. In this work, only a square lattice has been considered for both two- and three-dimensional simulations due to their ease of implementation, especially in three-dimensions, and their suitability for parallelisation. The configurations are denoted in the form $D_{x}Q_{y}$, where $x$ is the number of dimensions (two or three) and $y$ is the number of discrete particle distributions. For two-dimensions, a $D_{2}Q_{9}$ lattice has been widely used for flow simulations and was used here. In three-dimensions, a $D_{3}Q_{15}$ lattice was favoured, after consideration of $D_{3}Q_{15}$, $D_{3}Q_{19}$ and $D_{3}Q_{27}$ configurations, shown in section 3.2.2. Each of these configurations has a particle distribution set, $f_{\alpha}|_{\alpha=0,1,...,y-1}$, with a corresponding velocity vector $e_{\alpha}$. The equilibrium distribution for both the $D_{2}Q_{9}$ and $D_{3}Q_{15}$ models is:

$$f^{(eq)}_{\alpha} = w_{\alpha}\rho \left[ 1 + 3(e_{\alpha} \cdot u) + \frac{9}{2}(e_{\alpha} \cdot u)^{2} - \frac{3}{2}u^{2} \right]$$  \hspace{1cm} (3.1)$$

where $w_{\alpha}$ is the weighting factor for $f_{\alpha}$ and $c \equiv \frac{\delta}{\delta_{t}}$ is the unit speed in the model. The speed of sound in the model, $c_{s} = \frac{1}{\sqrt{3}}$. In all the models here $\delta_{x}$ is the lattice constant, $\delta_{t}$ is the time step and $\delta_{x} = \delta_{t} = 1$. All the configurations considered in this study have a rest particle, but this is not an essential feature of a LBM particle distribution.
3.2.1 Two Dimensions: The D2Q9 Lattice

This configuration uses nine particle distributions, one each for the eight nearest neighbours on the lattice and one for the rest particle. The velocity vectors for this model are:

$$e_\alpha = \begin{cases} 
(0,0), & \alpha = 0 \\
(\pm 1,0),(0,\pm 1), & \alpha = 1,2,3,4 \\
(\pm 1,\pm 1), & \alpha = 5,6,7,8 
\end{cases} \quad (3.2)$$

![Diagram of the D2Q9 lattice configuration](image-url)
and the values for the weighting factors are:

\[
    w_{\alpha} = \begin{cases} 
        \frac{4}{9}, & \alpha = 0 \\
        \frac{1}{9}, & \alpha = 1, 2, 3, 4 \\
        \frac{1}{36}, & \alpha = 5, 6, 7, 8 
    \end{cases}
\]  

(3.3)

### 3.2.2 Three Dimensional Lattices

For the three-dimensional lattice, there are some further complications. Due to the massive increase of computational power required for three-dimensional simulations, some compromise is need over the number of particle distributions in the simulation. The obvious extension from the two-dimensional simulation is to have a distribution for each of the nearest neighbours plus the rest distribution. This gives 26 for the neighbours plus the rest particle distribution, making a total of 27 distributions. This is therefore known as the D3Q27 model. For a 100 x 100 lattice, the move to three dimensions increases the number of lattice nodes by a factor of 100. When the extra demands of the D3Q27 model are included, this equates to an increase in computation of approximately 300 times. There are two other three-dimensional square lattice configurations: D3Q15 and D3Q19. The difference between these three models is the number of distributions on the links to the second nearest neighbour points on the lattice.
Figure 3.2: Layout of the discrete velocity vectors on the D3Q27 lattice configuration.

The D3Q27 Lattice

The D3Q27 lattice configuration includes a distribution for all of the 26 nearest neighbours of a given lattice point plus one for the rest particle. While this gives the most complete interaction between the lattice sites, it is computationally much more expensive than the other two other models considered, while the increased completeness in the fluid node interactions gives little extra accuracy.
to the results of the simulation. The discrete velocities for this model are:

\[ e_\alpha = \begin{cases} 
(0, 0, 0), & \alpha = 0 \\
(\pm 1, 0, 0), (0, \pm 1, 0), (0, 0, \pm 1), & \alpha = 1, 2, \ldots, 6 \\
(\pm 1, \pm 1, 0), (\pm 1, 0, \pm 1), (0, \pm 1, \pm 1), & \alpha = 7, 8, \ldots, 18 \\
(\pm 1, \pm 1, \pm 1), & \alpha = 19, 20, \ldots, 26 
\end{cases} \]  

(3.4)

and the values for the weighting factors are:

\[ w_\alpha = \begin{cases} 
\frac{8}{27}, & \alpha = 0 \\
\frac{2}{27}, & \alpha = 1, 2, \ldots, 6 \\
\frac{1}{54}, & \alpha = 7, 8, \ldots, 18 \\
\frac{1}{216}, & \alpha = 19, 20, \ldots, 26 
\end{cases} \]  

(3.5)

The D3Q19 Lattice

The D3Q19 lattice configuration significantly reduces the computational requirements from those of the D3Q27 model. However it is still more computationally intensive than the D3Q15 model while giving no real advantage in terms of accuracy in the model. The velocity vectors for this model are:

\[ e_\alpha = \begin{cases} 
(0, 0, 0), & \alpha = 0 \\
(\pm 1, 0, 0), (0, \pm 1, 0), (0, 0, \pm 1), & \alpha = 1, 2, \ldots, 6 \\
(\pm 1, \pm 1, 0), (\pm 1, 0, \pm 1), (0, \pm 1, \pm 1), & \alpha = 7, 8, \ldots, 18 
\end{cases} \]  

(3.6)
Chapter 3. The Lattice Boltzmann Model

Figure 3.3: Layout of the discrete velocity vectors on the D3Q19 lattice configuration.

and the values for the weighting factors are:

$$w_\alpha = \begin{cases} 
\frac{1}{3}, & \alpha = 0 \\
\frac{1}{18}, & \alpha = 1, 2, \ldots, 6 \\
\frac{1}{36}, & \alpha = 7, 8, \ldots, 18 
\end{cases} \quad (3.7)$$
The D3Q15 Lattice

The D3Q15 lattice provides the most computationally efficient solution. The further loss of the links to the diagonal nearest neighbours does not significantly affect the results generated while reducing the amount of computation required by the D3Q27 model by almost a factor of two. Due to the extremely large number of timesteps required to achieve turbulence in oscillatory flows using the LBM, this saving can reduce the amount of time taken for simulations to run by weeks or months, making this is a very significant saving. The velocity vectors

Figure 3.4: Layout of the discrete velocity vectors on the D3Q15 lattice configuration.
Chapter 3. The Lattice Boltzmann Model

for this model are:

$$e_\alpha = \begin{cases} 
(0, 0, 0), & \alpha = 0 \\
(\pm 1, 0, 0), (0, \pm 1, 0), (0, 0, \pm 1), & \alpha = 1, 2, \ldots, 6 \\
(\pm 1, \pm 1, \pm 1), & \alpha = 7, 8, \ldots, 15 
\end{cases} \quad (3.8)$$

and the values for the weighting factors are:

$$w_\alpha = \begin{cases} 
\frac{2}{9}, & \alpha = 0 \\
\frac{1}{9}, & \alpha = 1, 2, \ldots, 6 \\
\frac{1}{72}, & \alpha = 7, 8, \ldots, 15 
\end{cases} \quad (3.9)$$

3.3 The Lattice Boltzmann Equation

The lattice Boltzmann equation (LBE) is comparable to the classic Boltzmann equation. It solves the microscopic kinetic equation for particle distributions, $f(r, t)$, where $r$ is the position in the fluid and $t$ is the time. The LBE is given by:

$$f_\alpha(r + e_\alpha, t + 1) - f_\alpha(r, t) = \Omega_\alpha(f) \quad (3.10)$$

Here, $\Omega_\alpha$ is the collision function. The macroscopic variables density, $\rho$, and velocity, $u$, can be recovered from the particle distributions as follows:

$$\rho(r, t) = \sum_\alpha f_\alpha(r, t) \quad (3.11)$$
The LBE breaks down into two distinct steps when it is implemented. These are the collision and the streaming. The collision involves the calculation of the distributions for the new timestep, \( t + 1 \), by relaxing the distributions at the current time, \( t \), toward the equilibrium distribution functions \( \bar{f} \). The streaming distributes these new distributions to the correct lattice sites according to a set of rules for each lattice site. These rules depend on the location of the lattice site, whether it is on a boundary site or a fluid site. If the site is a boundary site, the rules also change with the orientation of the wall and also whether it is a flat wall or a corner, etc.

### 3.3.1 The Collision Function

The collision operator given in equation 3.10 is a simplified version of the lattice gas collision function, \( \langle \Omega_\alpha(n) \rangle \)[34]. While the use of the LBE eliminates the statistical noise of the LGCA, when the collision operator \( \Omega_\alpha(f) = \langle \Omega_\alpha(n) \rangle \), it still depends on \( 2^m \) Boolean input and output states, where \( m \) is the number of distributions in the model. For a D3Q27 model, this leads to a \( 2^{27} \times 2^{27} \) collision matrix. While the computational overhead is less for other models, it can still be a limiting factor. This matrix can be reduced to a \( m \times m \) matrix, by expanding the distribution function:

\[
    f_\alpha = f_\alpha^{(eq)} + f_\alpha^*
\]  

(3.13)
Chapter 3. The Lattice Boltzmann Model

where \( f^{(eq)}_{\alpha} \) is the equilibrium part of the distribution function and \( f^*_{\alpha} \ll f^{(eq)}_{\alpha} \) is the non-equilibrium part. When the collision operator is expanded about the equilibrium distribution, this gives

\[
\Omega_{\alpha}(f_{\alpha}) = \Omega_{\alpha}(f^{(eq)}_{\alpha}) + \sum_{\beta} f^*_{\beta} \Omega_{\alpha\beta} + O((f^*)^2) \tag{3.14}
\]

As \( \Omega_{\alpha}(f^{(eq)}_{\alpha}) = 0 \), this allows that lattice Boltzmann equation to be written as

\[
f_{\alpha}(r + e_{\alpha}, t + 1) - f_{\alpha}(r, t) = \sum_{\beta} \Omega_{\alpha\beta}(f_{\beta} - f_{\beta}^{eq}) \tag{3.15}
\]

The collision operator can be further simplified[30] by assuming that the particle distribution functions relax towards their equilibrium states at a constant rate

\[
\frac{\partial \Omega_{\alpha}}{\partial f_{\beta}} \bigg|_{f_{\beta}=f_{\beta}^{eq}} = -\frac{1}{\tau} \delta_{\alpha\beta} \tag{3.16}
\]

This gives a collision term of

\[
\Omega_{\alpha} = -\frac{1}{\tau} (f_{\alpha} - f_{\alpha}^{eq}) \tag{3.17}
\]

When the collision operator is in this form, the lattice Boltzmann equation is known as the lattice BGK equation, due to this similarity of this operator to the
classic Boltzmann BGK operator\[35\]. The lattice BGK equation has the form:

\[
f_a(r + e_a, t + 1) - f_a(r, t) = \Omega_a(f_a(r, t))
\]  

(3.18)

### 3.3.2 Streaming

The LBM contains two separate steps. The first, which has been discussed in the previous section is the Collision function which changes the relative sizes of the distribution functions between the particle distributions. The second step is the streaming. This is where the neighbouring nodes on the lattice exchange distribution functions along links according to the lattice configuration. For each of the velocity vectors described in section 3.2, each distribution is streamed to the neighbouring site in that direction. So for the third distribution on the D2Q9 lattice, \(f_3(r, t)\) where \(r = (i, j)\) and \(e_3 = (-1, 0)\), is moved to \(f_3(r', t)\) where \(r' = r + e_3 = (i - 1, j)\). More generally, this means that a given distribution, \(f_a(r, t)\) is moved to \(f_a(r', t)\), where \(r' = r + e_a\). The streaming rules described here are for fluids sites in the simulation, that is a site where none of the particle distributions would stream into a boundary. The streaming rules are different for sites where the particle distributions do stream into a boundary. These differences are discussed in section 3.4.
3.4 Boundary Conditions

In the LBM, the boundary is a set of nodes where the streaming rules (section 3.3.2) are different than those fluid sites in such a way as to erect a solid boundary in the fluid simulation. There are many different ways that this can be implemented in the LBM. These different models all have different computational demands and some have different levels of accuracy. Some of the more noted ones are by Inamuro[36], Chen[37], Maier[38], Zou[39] and Noble[40]. The Halfway Bounce Back boundary, section 3.4.1, was favoured for two main reasons. First, it is a simple boundary to implement and is therefore computationally efficient. The second is that when a Halfway Bounce Back boundary is parallel to main lattice links, a 2nd order accuracy is achieved at no extra computational cost[37, 41].

3.4.1 Halfway Bounce Back Boundary

The halfway bounce back boundary places the boundary of the flow halfway between two lattice nodes. This positioning allows the accuracy of this boundary condition to be improved for no extra computational cost, as long as the boundary is parallel to the main lattice links. This boundary condition works in a very similar way to the normal bounce back condition. Figure 3.5(a) highlights two particle distributions at some timestep \( t \). These then stream to the positions
Figure 3.5: Graphical representation of two sample particle distributions, coloured red and blue, streaming to the boundary and returning with the halfway bounce back boundary condition. (a) shows the position of the two distributions at timestep $t$. (b) shows the positions at timestep $t+1$ as they approach the boundary. (c) shows them after bouncing off the boundary back in the direction they came from.
shown in figure 3.5(b) at timestep $t + 1$. When the particle distributions are streamed at the next timesteps, they are moved to the opposite distribution on the same lattice node. For the particle distribution coloured blue, this means that it is moved from the particle distribution numbered 4 in figure 3.1 to that numbered 2. This corresponds to a streaming to the boundary, which is half a lattice unit of $\delta_z$, and then returning in the opposite direction, again half a lattice unit of $\delta_z$. This means that the particle distribution has streamed a length of $\delta_z$, which is the same distance a particle distribution at a fluid node would stream.

### 3.5 Driving the Oscillation

The oscillations of the fluid in the LBM were driven using a body force:

$$ F = \begin{pmatrix} P_{GRAD} \cos(\omega t) \\ 0 \\ 0 \end{pmatrix} $$

(3.19)

The third term is obviously unnecessary in two-dimensional simulations. The forcing term is inserted into the lattice-Boltzmann equation as an addition to the collision function so that equation 3.18 becomes:

$$ f_\alpha(r + r_\alpha, t + 1) - f_\alpha(r, t) = \Omega_\alpha(f_\alpha(r, t)) + 3 \frac{2\tau - 1}{2\tau} \omega_\alpha \mathbf{F}_\alpha $$

(3.20)
Chapter 3. The Lattice Boltzmann Model

Equation 3.12 is redefined to give a fluids velocity\cite{42} of:

\[
\rho(r, t) \mathbf{u}(r, t) = \sum_a e_a f_a(r, t) + \frac{F}{2} \tag{3.21}
\]

This body force can be shown to satisfy the Navier-Stokes and continuity equations up to second order accuracy\cite{42}. The body force is applied to the whole of the lattice, which means that all the points in the lattice are at the same phase in the oscillation at any given time. The lattice-Boltzmann model is only approximately incompressible due to non-local interactions\cite{43, 44}, so if the fluid were only driven at one plane, the planes of the lattice could become slightly out of phase with each other, which could affect the phase at which transition to turbulence occurs. Several other schemes have been put forward to impose a body force on the LBM, most notably by Guo et al\cite{45}. Although the body force proposed by Guo et al results in a smaller error than that used here, its increased complexity and especially the introduction of non-local terms in the calculation of the body force makes it unsuitable for use in these simulations. In parallel computation, detailed in section 3.9, non-local computations require an increase the size of HALO regions, which in turn slow the computations considerably.
3.6 Initialising the Laminar Regime

The calculated solutions to the Navier-Stokes equation detailed in sections 2.2.1 and 2.2.2 can be used to measure the accuracy of the implemented computational solutions. They can also be used to cut down the simulation times when modelling turbulent flows, as it becomes possible to initialise the simulation to a fully developed laminar velocity profile rather than having to run the simulation for a large number of oscillations until the profile is fully developed. When a simulation is started from a zero velocity profile, $u(r) = 0, \forall r$, it takes $\sim 30$ oscillations for the velocity profile to closely approach the analytical solution for this situation. This can be eliminated by using the analytical solution as a starting point for the simulations. Thus only a small number of timesteps are required for the simulation velocity profile to approach the analytical one enabling a large reduction in simulation run times.

This solution given in section 2.2.1 is also used to initialise the three-dimensional channel simulations, as the laminar velocity profile for flow between two infinitely large parallel plates is the same as that for two-dimensional channel flow.

3.7 Triggering the Turbulent Regime

Due to the symmetry and high precision of computer simulations, it is possible to increase the Reynolds number of a flow in the LBM well into a turbulent
regime without turbulence occurring. This is caused by the walls of the channel or tube being exactly smooth and so the velocity profile which forms is exactly symmetric. In an experimental situation, the walls of the tube would not be exactly smooth because of surface variations in the material used. Even if the material used were a smooth as possible, variations caused by the molecular makeup would mean that the velocity profile would not be exactly symmetric. For turbulence to be triggered, this symmetry has to be broken. This was achieved following Akhavan by the introduction of a small random perturbation into the particle distributions after the application of the collision function, but before the streaming. The symmetry of the flow is disturbed enough by this to allow for turbulence to occur in a flow with a sufficiently high Reynolds number. In this study, the random perturbations were added to all sites on the lattice. Several test simulations showed that the location at which the perturbations were added influenced whether turbulence was generated. If the perturbations were only added around the centre of the channel, then no turbulence was generated in the flow. Conversely, if the perturbations were added only in the Stokes boundary layer, turbulence was generated. However, for all the simulations this study, the perturbations were added at all points on the lattice to give a uniform flow.
Chapter 3. The Lattice Boltzmann Model

3.8 Flow Domains

The domain for the two-dimensional simulations is shown in figure 3.6. There are two directions, the streamwise direction (labelled $x$) and the spanwise direction (labelled $y$). The flow oscillates in the $x$-direction, with periodic boundary conditions. The channel is bounded by walls at the edges of the spanwise direction. The basic domain for three-dimensional channel simulations, three-dimensional duct simulations and PIV measurements is shown in figure 3.7. The third dimension is called the cross-stream direction (labelled $z$). There are slight differences in the configuration of the three-dimensional domain for each of these different situations. As in the two-dimensional simulations, walls are implemented at the edges of the spanwise direction. For the three-dimensional channel and three-dimensional duct simulations, periodic boundary conditions are implemented at the edges of the streamwise direction. In the PIV measurements, the
edges of the streamwise direction mark the edge of the camera range, but the duct carries on to the motor driving the flow. In the three-dimensional channel simulations, a periodic boundary condition is implemented at the edges of the $z$-direction to create an infinite channel. For the three-dimensional simulations and PIV measurements, there are walls at the edges of the cross-stream direction to create a square duct for the flow. In all diagrams, the size of the domains in each direction have been normalised to the half width of the spanwise direction, $r$. The spanwise direction has also been centred at $y = 0$, so that it runs in the range $-1 < y < 1$, with the boundaries occurring at $y = \pm 1$. For the three-dimensional duct simulation and the PIV measurements, the cross-stream direction is the same size and configuration as the spanwise direction, as the flow here is in a square duct. Therefore in all diagrams for these configurations, the $z$-component is centred on zero and runs in the range $-1 < z < 1$.

### 3.9 Parallel Implementation of the Model

The size of the computational domain and the number of timesteps required to achieve a turbulent flow using the lattice-Boltzmann method mean that the only way to obtain results in a sensible time frame, particularly for three-dimensional simulations, is to run the code on a parallel computer.
Figure 3.7: Diagram of the computational domain for the three-dimensional channel simulations, three-dimensional simulations and PIV measurements.
There are two main types of parallel computer in current use. These are shared memory and distributed-memory machines. Shared memory machines are highly specialised and expensive. The multiple processors they use all share the same system RAM. The most common programming standard used with this type of machine is OpenMP [46], which is relatively simple to implement for most problems.

On a distributed-memory machine, each processor has its own separate area of RAM which the other processors cannot access directly. This makes distributed-memory machines unsuitable for problems with long-range interactions. They are, however much more scalable than shared memory machines which start to run into memory bandwidth problems if they have too many processors. Distributed-memory memory machines come in two types. The first comprises shared memory machines, being specially built and expensive computers. The other type is known as Beowulf[47] and uses 'off the shelf' components to create cheap parallel computers. The main software standard for distributed-memory machines is the Message Passing Interface (MPI) standard[48], although there are others. The lattice-Boltzmann method lends itself to distributed-memory parallel computing as all the interactions between the lattice sites are local (or nearest neighbour) in nature with no long distance interactions.
3.9.1 Parallel Computer Choices

There are many factors which need to be considered when choosing the parallel computer to use for a large computation. The wrong choice could vastly affect the length of simulation runs and therefore the quality of results which can be obtained in a given time span. For this project, there were two main options. The first was to use a specially built, shared memory computer available from EPCC[49]. These computers are capable of using both the OpenMP and MPI parallelisation standards. The other option is to construct a Beowulf cluster specifically for this computation. A Beowulf system would only be able to use the MPI standard, as the memory is distributed in the different machines. The main considerations when deciding which systems to use are the size of the lattices and the number of timesteps required in each simulation. As memory is cheap and plentiful, the over riding consideration is the number of timesteps per oscillation required. To generate a flow with the Reynolds number required for turbulence to occur using a lattice-Boltzmann model requires 1,000,000 timesteps per oscillation. This means that each simulation will have a very long run time. When considering distributed memory parallel computers, an important factor is the number of computing nodes on which a simulation is to be run compared to the size of the computational domain. In distributed memory parallel computing, there are three important components of the run time. These could be termed the serial, parallel and communication components. The serial component is the
run time which cannot be reduced by splitting the computation over multiple nodes and occurs mainly during the initialisation and data output sections of the program. The parallel component contains the bulk of the computational load and occurs during the main parts of the simulation. The communication component involves the communication of information, such as neighbouring data point values, between the different nodes in the parallel computer. The run time of this component cannot be significantly reduced by the inclusion of more nodes and is sometimes increased by this. In a lattice-Boltzmann simulation, each timestep constitutes one section of communication, which takes an approximately fixed amount of time. Spreading a computation over more nodes would reduce the overall time taken for each timestep, but the time for the communication would remain constant. Splitting the domain over more and more nodes would eventually lead to this communication time becoming the dominant part of the simulation's run time and any further reductions in runtime would be negligible. With this in mind, the large number of nodes available in the clusters at EPCC would do little to help speed up the simulations. Also, the run time of simulations on the EPCC clusters is limited which would be very restrictive on this kind of simulation. With this in mind, it was decided to construct a Beowulf cluster for the simulations, with emphasis on the raw processor and networking speed to achieve the shortest run times.

All three-dimensional simulations were carried out on a Beowulf cluster of eight custom built IBM PC compatible machines. Each machine was equipped
with a 2.56\,GHz Intel Pentium 4 CPU and 256\,MB of RAM. The machines were running the Redhat Linux operating system with the LAM/MPI MPI software\cite{50}. The machines were connected using a gigabit ethernet connection. This cluster was used for all the three-dimensional simulations in this study cost \( \sim £4000 \). This bought the components for eight PC’s and the gigabit ethernet switch. The PC’s were then self-assembled. Each machine contained only the minimum of components to keep the cost down and also try to improved the performance of the cluster by reducing the demands on the system from components that aren’t essential for numerical simulation work.

As the lattice-Boltzmann method uses only nearest neighbour interactions, it scales well if further nodes added into the cluster correspond to an increase the lattice size. However the benefits splitting the same size lattice of more nodes are not as great, as this increases the communication time as an overall proportion of the runtime. This effect can be minimised by the choice of domain decomposition, shown in section 3.9.1

Splitting the Computational Domain

There are many ways to split the computational domain in parallel computing applications. These include splits in either space or time, or simultaneous running of multiple simulations in statistical situations. As most turbulence parameters are highly statistical in nature, it would seem that better results could
be gained by duplicating the simulations. However the run times, particularly for three-dimensional simulations, are too prohibitive with current equipment. Also, due to the deterministic nature of lattice-Boltzmann simulations, splitting across time is not possible and so a spatial split is the only realistic option.

There are many different ways to split the computational domain in space. For a lattice-Boltzmann simulation, there are two factors. The first of these is the ratio of inter-facial surface area to volume of the sub-domains. It is essential to minimise this ratio as communication between the nodes in the simulation is much slower than in the lattice-Boltzmann computation. The inter-facial surface area is the number of nodes in the HALO regions at the edge of the sub-domain, which stores a duplicate of the lattice nodes at the edge of the neighbouring sub-domains. This allows the nodes at the edge of the sub-domain to be updated. Transferring this information between sub-domains is much slower on a per node basis than computing the lattice-Boltzmann update at any one timestep. The second is the complexity of the computational domain in the simulation. Depending on the the shape of the fluid domain being simulated, it may be very complex to split the domain. For example, in a system of pipes with many junctions, the simplest way is to place the system in a large square box and then split the box between the processors. This leaves large areas of empty space which reduces the efficiency of the simulation. A more effective way is to split the lattice along the pipe. This causes problems with the calculation of the neighbouring processors and the exact location of the interfaces on each processor.
All the simulations considered here are of straight sections of pipe, either channels in two-dimensions or square ducts in three-dimensions. This greatly simplifies the splitting of the computational domain. The most simple to implement is cutting the pipe into sections with the cuts being perpendicular to the streamwise direction, as shown in figure 3.8. While this is not the most efficient in terms of minimising the ratio of inter-facial surface area to sub-domain volume, it does provide advantages in terms of coding simplicity and also flexibility in the number of processors which can be used on the domain. While the domain decomposition shown in figure 3.9 improves the inter-facial surface area to sub-domain volume ratio, its more complex structure means that it has to be run on a multiple of four processors. This reduces flexibility in options for execution on parallel systems with different numbers of processors. It also provides a more complex coding
Figure 3.9: Diagram of the three-dimensional computational domain, indicating the processor boundaries for an eight processor split.

challenge, which means that the code must be edited every time the program is run on a different number of processors. With the slicing method depicted in figure 3.8, only very simple changes to the length of each sub-domain are required, rather than a full reorganisation of neighbouring processors.
Chapter 4

Particle Image Velocimetry Method

The measurement of velocity flow fields is a very important area of fluid dynamics in both Physics and Engineering. For this work, there are two principal requirements. Any technique must be non-intrusive so that the flow is not disturbed by the measuring technique. It is also important to be able to view a large flow field so that flow patterns on many different length scales can be observed. Particle Image Velocimetry (PIV) was chosen for this study as it complies with both of these requirements.
4.1 PIV Background

PIV has been developed as a technique over many years. The development of the laser at the end of the 1960's was the first step, as it allows the generation of the very short, very intense pulses of light that are required for PIV. The technique has been gradually developed over the years since then. In the last 10 years, the advent of digital photography and advances in computer technology have greatly increased the simplicity of the measuring process and the accuracy and range of velocities and flow regimes which can be measured using PIV. PIV techniques have been well documented over the years of its development. Reviews of the developments in PIV can be found in papers by Adrian[51] and by Lauterborn and Vogel[52]. Further information can be found in two text books specifically about PIV techniques by Westerweel[53] and by Raffel et al.[54]. A collection of papers covering the entire process of PIV from taking the measurements to analysis of the results has also been compiled by Grant[55].

4.2 PIV Technique

PIV is a non-invasive flow visualisation technique which uses seeding particles moving in the flow. These are illuminated with a laser instantaneously and photographed at a time $t$. This is then repeated a short time later, $t + \delta t$, and a cross-correlation is performed on the two images to give a velocity map of the flow
in the fluid. It is also possible to use two exposures on the same image and then perform an auto-correlation. These two steps in are known as Image Acquisition and Image Analysis.

4.3 Image Acquisition

The image acquisition part of PIV is where the images of the seeding particles moving with the flow are captured. This involves the illumination of the flow, the timing of the gap between the two images and also the camera used in the set up. These features are described in this section, as are details of the method used to drive the flow and the seeding particles used.

4.3.1 Flow Illumination

PIV requires fast, short and intense bursts of light, so a laser is almost always used to illuminate the flow. For accurate results, it is important that the particles do not move too far between pulses, so for high velocity measurements, very fast pulse rates are required. There must also be sufficient intensity of light so that the particles scatter enough light to the camera for an accurate measurement to be made. Pulsed lasers, such as Nd.Yag or Copper Vapour lasers can generate the high intensity pulses required to measure velocities right up to the supersonic range ($\sim 600ms^{-1}$)
Chapter 4. Particle Image Velocimetry Method

The Copper Vapour Laser

For these measurements, an Oxford Lasers LS 20-50 Copper Vapour laser was used. This is a Class IV laser and provides a very powerful beam which can be used in either continuous or pulsed modes. In pulsed mode, it can pulse with a frequency of up to $50kHz$, which is far in excess of the requirements of this experiment. The duration of each light pulse is $\sim 5-60ns$ which is a short enough time for the particles in the fluid to be effectively stationary in the images taken for the PIV measurements. The laser produces light at two wavelengths which are $510.6nm$ and $578.2nm$. The light is generated from the excitement of copper atoms.

Laser Beam Optics

To make the measurements, the light from the laser needs to be focused into a thin and even sheet across the area of interest in the fluid. The beam is focused through several lenses to achieve this. There are, however, two main problems. The first is the width of the light sheet. This restricts the area which can be investigated. The second is that the intensity of the light sheet has a Gaussian profile which means that the intensity is significantly less at the edges of the light sheet. This reduces the accuracy of reading which can be obtained in areas illuminated with the edges of the light sheet, which in turn reduces further the area which can be studied.
4.3.2 Camera

The images in this experiment were captured using a SensiCam Double Shutter PCO digital camera. This allowed the capture of two images with a suitable time gap between them to give good PIV results in this application. The camera has several different modes for recording images. For these experiments, the Double Long mode was used. In this mode, the camera takes two images, separated by a time of $\sim 1\mu s$. The exposure time of the first images is controlled by the length of the pulse sent to the camera. After this, there is a delay of $\sim 1\mu s$ before the second exposure. The length second exposure is the time taken for the first image to download to the computer, which takes $\sim 250ms$.

4.3.3 Fluid

As the PIV images are to be taken through a perspex tube, it is important to match the refractive index of the fluid to that of the tubing to eliminate refraction in the measurements. The refractive index of perspex is $\sim 1.49[56]$. As the refractive index of water is only $\sim 1.33$, it is clearly not suitable for this experiment. After tests with several solutions, including sodium-chloride solution and ammonium-thiocyanate solution, sodium-iodide was chosen. Sodium-iodide ($NaI$) solution provides a clear fluid with the required refractive index. A solution of 62% – 64% $NaI$ by weight with deionized water provides a solution with a refractive index $\sim 1.49$. This solution also has a low kinematic viscosity, $\nu \sim$
1.1 \times 10^{-6} m^2 s^{-1}, which allows the generation of high Reynolds number flows fairly easily. There are some problems with this solution. The refractive index of the solution does vary slightly with wavelength of light and temperature, but these variations are small in the ranges considered in this experiment. Also the experimental setup used means that most of the light moving through the perspex does so at or close to 90°, which minimises the effects of refraction. This solution also deteriorates over time as a minute number of I$_3^-$ ions form in the fluid. These absorb light in certain ranges of the visible spectrum and so limit the effectiveness of the solution. This can be reduced by limiting the exposure of the solution to light and oxygen. The reduction of transmission of the NaI solution is particularly severe for the frequency doubled Nd:YAG laser used in the experiments by Uzol et al[57] which has a wavelength of 532nm. The Copper Vapour laser used in these experiments produces light at two wavelengths, 510.6nm and 578.2nm. While the first wavelength is affected to a greater extent, as shown in [57], figure 5, the reduction in transmission for the second wavelength is much less significant. To reduce transmission deterioration, the solution was kept in a light free environment. It was not possible to keep the solution in an oxygen free environment, as suggested by Uzol et al, but it was kept in an air tight container when not in use. As an additional precaution and due to the small amounts of fluid required, \~ 750ml, it was possible to make a new solution every few days to further minimise the effects of the I$_3^-$ ion formation.
4.3.4 Seeding Particles

The seeding particles used in these experiments were silver coated glass spheres, with a size of $10\mu m$. The use of charged polystyrene spheres was considered, but they tended to clump together in the fluid, probably because of the solution's high salt content. The glass spheres appeared to be slightly less dense than the NaI solution. After approximately an hour, they had mostly floated to the surface of the NaI solution. It was therefore important to make sure that they were evenly dispersed throughout the fluid before any measurements were taken.

4.3.5 Driving the Flow

The fluid was driven by a Micropump pump controlled by a PC using the LABView software. The software was used to generate a sine wave at a frequency, $f = 0.5Hz$.

4.3.6 Timings

The timings of the laser pulses and the control of the camera were controlled by signals generated by a BNC 500 pulse generator modified with an External Burst Mode. This additional mode allows the box to produce multiple pulses from one output in a single cycle, while a standard BNC 500 pulse generator can only produce one pulse per output per cycle. This was in turn controlled by a
delay box, which provided a pulse to trigger the BNC sequence. The delay box was triggered when the sine wave signal changed from negative to positive. It then sent a pulse to the BNC box with a delay which could be set from 0s to 1s in 1ms steps. This was increased in stages of 0.25s to give readings at phase intervals of 45°. As the maximum delay of this box was 1s, this only allowed measurements to be made for the first half of the period. For the second half, the delays in the BNC box were all increased by 1s. The BNC box then generated pulses which controlled when the laser fired and when the camera exposed. The exact timings are shown in figure 4.1. It should be noted that a delay of 0s on

Figure 4.1: Timeline for PIV experiment. Note that timeline is not to scale.

the delay box produced a PIV measurement for a phase of 45°, 0.25s for a phase of 90°, etc. The measurement for a phase of 0° was produced with a delay of
1.75s, which is equivalent to a phase of 360°. The full settings on the BNC signal generator are shown in table 4.1.

<table>
<thead>
<tr>
<th></th>
<th>Channel T1</th>
<th>Channel T2</th>
<th>Channel T3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polarity</td>
<td>Positive</td>
<td>Negative</td>
<td>Positive</td>
</tr>
<tr>
<td>Width (s)</td>
<td>0.00000200</td>
<td>0.35000000</td>
<td>0.00056000</td>
</tr>
<tr>
<td>Delay (s)</td>
<td>0.24994000</td>
<td>0.24000000</td>
<td>0.24950000</td>
</tr>
<tr>
<td>Number of Bursts</td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mode</td>
<td>ExtBurst</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total Period (s)</td>
<td>0.00000800</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4.1: Full setting for BNC 500 signal generator.

4.3.7 Image Capture

The images taken by the camera, are captured using a computer running the SensiCam package. Due to the memory limitations of the computer, there is a limit on the number of exposures which can be stored at any one time. It is possible to store up to to ~ 20 exposures in the memory, although this varies by ±2 images, depending on the density of particles and the amount of glare. The two exposures of the camera are stored “one on top of the other” in bitmap form. The stored image is split in to the two exposures and, by framing the edges to remove sections which have captured the perspex duct, made ready for analysis.
Figure 4.2: Schematic diagram of the Image Acquisition setup.

4.3.8 Experimental Setup

Figure 4.2 shows a schematic diagram of the setup used to acquire the images for the PIV measurements. The fluid was driven in the duct by the motor, which was controlled by the sine wave generated by the LABView computer. This sine wave also triggered the delay box, which in turn triggered the BNC Pulse Generator. The BNC Pulse Generator co-ordinated the PCO camera and the laser in recording the images. These were then stored on the PCO Camera computer.
4.4 Image Analysis

Image analysis involves performing statistical analysis of the images obtained using the techniques described in section 4.3. Each of the images is divided into a grid of smaller sections and then a statistical correlation is applied to obtain the average displacement of the seeding particles in this area. The displacement is then converted to a velocity vector. This gives a velocity map of the whole image. As the particle displacements are averaged in each integration area, the size of the areas limits the spatial resolution of the velocity map which can be obtained. A typical size for these areas is $32 \times 32$ pixels, which on an image of $1024 \times 1024$ pixels gives $32 \times 32$ vectors. All of the analysis for PIV in this thesis has been carried out using the TedPIV software, which was developed at the University of Edinburgh by Ted Schlicke[58].

4.4.1 Cross-correlation

There are two types of analysis for PIV images to obtain the displacement information. For images where there is a double exposure on one image, an autocorrelation is performed. Where there are two separate images, a cross-correlation is performed. This is the case for all of the measurements in this thesis. If the seeding particles visible on each of the images, one and two, are described by the functions $f(x, y)$ and $g(x, y)$, then the cross-correlation, $C_{fg}(x, y)$ is given
Chapter 4. Particle Image Velocimetry Method

as:

\[ C_{fg}(x, y) = f(x, y) \otimes g(x, y) = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} f(m, n)g^*(m - x, n - y)dmdn \]  \hspace{1cm} (4.1)

where \( \otimes \) represents cross-correlation and \( g^*(x, y) \) is the complex conjugate of \( g(x, y) \). For digital images, the information is discretised and equation 4.1 becomes:

\[ C_{fg}(x, y) = \sum_{m=0}^{M} \sum_{n=0}^{N} f(m, n)g^*(m - x, n - y) \]  \hspace{1cm} (4.2)

It is also possible to calculate the cross-correlation using the Fourier transforms of \( f(x, y) \) and \( g(x, y) \):

\[ C_{fg}(x, y) = f(x, y) \otimes g(x, y) \Leftrightarrow F(C_{fg}(x, y)) = F(u, v)G^*(u, v) \]  \hspace{1cm} (4.3)

where \( F \) denotes a Fourier transform and \( F(u, v) \) and \( G^*(u, v) \) are the Fourier transforms of \( f(x, y) \) and \( g^*(x, y) \). This allows the cross-correlation shown in equation 4.1 to be computed by multiplying Fourier transforms of \( f(x, y) \) and \( g(x, y) \), and then taking the inverse Fourier transform of the result. This is a much quicker way to compute the cross-correlation[59], helped largely by the availability of very efficient Fourier transform routines.
4.4.2 Peak Detection

One of the most important functions of the PIV analysis program is locating the correlation peaks, which in turns allows the measurement of the particle displacement. As the images are stored in a digital format, it is only possible to measure the displacement to an accuracy of ±0.5 pixels. While this is fine for large displacement, in integration areas where there is a smaller displacement this becomes a problem. This can occur where there is a low velocity in the flow, such as in an area of turbulent flow or near the boundary of the flow channel. As this is a very important area of this study, it is important to increase the accuracy with which the particle displacement is measured. Measuring the location of the peak to sub-pixel accuracy can be achieved by applying a peak-fitting function to the nearest neighbouring points of the peak. There are two such schemes widely in use. These are parabolic-fit and Gaussian-fit estimators, both of which are referred to as three-point estimators[59]. It has been established that, in general, the Gaussian-fit is the most accurate at providing sub-pixel measurements. This is because particle intensities have a broadly Gaussian profile, so correlating two of these gives a correlation peak with a similar Gaussian profile. The Ted-PIV software used here implements a Gaussian-fit estimator which gives a pixel displacement accuracy of 0.1 ± 0.05 pixels[54].
4.4.3 Image Analysis Errors

It is clearly desirable to obtain the most accurate velocities from the PIV images once they have been collected. There are several areas where errors can be introduced into the analysis and also several things which can be done to increase the accuracy of the vector maps produced. All of these combine to affect the size of the correlation peaks given by the analysis. To make the velocities obtained more accurate, it is necessary to maximise the particle displacement correlation peaks compared to the random correlations.

Motion into and out of the plane

The correlation can be reduced by the movement of seeding particles into and out of the light sheet between the two exposures. This causes problems as particles can change size, as they move in and out of focus, or even appear or disappear completely between the two frames. This effect can be reduced by keeping $\delta_t$ as small as possible, ensuring that the majority of particles remain in the plane being studied for both of the camera exposures. There is a trade-off here, as $\delta_t$ must be kept large enough to allow the particles to move far enough for the displacement to be measured. For a given velocity, it is possible to calculate the maximum value of $\delta_t$ before the particles start to travel too far for the correlation, because it is impossible to correlate a particle between two images if it moves more than
50% of the integration width[60]. This leads to the restriction:

$$\delta_t \leq \frac{LI}{2\sigma v_0}$$  \hspace{1cm} (4.4)$$

where \( L \) is the width of the field of view, \( I \) is the integration size, \( \sigma \) is the resolution of the image and \( v_0 \) is the maximum flow velocity.

Concentration of Seeding Particles

It is important to have the correct concentration of seeding particles. If there are too few, there will be not enough correlation pairs to provide an accurate picture of what the flow is doing in a given integration area. The more particles there are, the higher the signal to noise ratio of the images and so the better the cross-correlation. However, if the particle density is too high, particles can start to overlap. This distorts the correlation peak and reduces the accuracy of the measurement. It has been established that the optimum number of particles is 7 per integration area. This gives a 95% valid detection rate[61]. As this is a statistical analysis, it would be ideal if the particles were evenly distributed over the integration area. Although this is not possible in practice, the distribution of particles can be improved by making sure that the fluid is throughly disturbed before any measurements are taken.
Chapter 4. Particle Image Velocimetry Method

Particle Image Size

The size of the particle image is also important to the correlation-peak size. A particle image that is too small gives a very narrow correlation peak. This reduces the values at neighbouring points to the peak and so the Gaussian-fit estimator will yield a displacement measurement which will tend toward an integer value in pixel units. This is known as peak-locking. If the particle image is too large, then the correlation peak will widen, decreasing the signal to noise ratio and increasing the likelihood of a non-sensible displacement vector. A particle size of approximately 2 pixels[54] has been shown to be the optimum size for cross-correlation. If the size of the particle image is less than 2 pixels, peak-locking can be avoided by slightly defocusing the image.

Integration Area Size

When measuring velocities with PIV, it is important that the size of the integration areas is chosen correctly. If the areas are too small, then there will not be sufficient particle pairs to give a good correlation. If the area is too large, then small scale information is lost due to averaging over the integration area. Coupled to this is the size of the velocity gradient over the integration area. A large gradient increases the width and decreases the height of the correlation peak, which results in reduced accuracy in the velocities detected. The correlation produces a bias, known as bias error, toward smaller velocities. This is caused
Chapter 4. Particle Image Velocimetry Method

by the size of the integration area and weights the results of the correlation by a window function. For example, the correlation of two identically sized integration areas is weighted by a triangular function. This has two effects on the outcome of the correlation: it broadens and reduces the height of the peak and it reduces the measured displacement. The effect is further increased by velocity gradients in the integration area, which is known as gradient bias. For an integration area of $32 \times 32$ pixels, the typical value for this error is 0.1 pixels, which is very significant for small displacements. This error can be reduced in a number of ways. If the size of the integration areas is varied between the two images, this reduces the error. The form of the weighting function is known, so it can be corrected with a weighting kernel. The bias error can also be reduced by re-integration, as discussed in section 4.4.4.

4.4.4 Improving Correlation

Re-integration

This is a method for reducing the errors in the measurement of displacement in PIV images. It was developed from observations by Westerweel et al.[62] who showed that the RMS error in the measured displacement decreases rapidly for displacements of less than ±0.5 pixels. There are three steps to this method. The first is to perform the cross-correlation as normal. The next is to shift the integration area of the first image by the nearest integer value of the displace-
ment. This will reduce the new correlation peak to less than ±0.5 pixels. If the cross-correlation is now repeated, it will give a greatly reduced error in the final displacement value. This also has the advantage of reducing the particle displacement which increases the number of correlation pairs, further increasing the signal to noise ratio and therefore the accuracy of the measurement. The increase in the signal to noise ratio allows a reduction in the size of the integration area, which increase the number of vectors which can be obtained from a given image pair.

Vector validation

The vectors produced from PIV images will not all be sensible. There are many reasons why a result would be considered invalid. These include glare from the fluid container surfaces, low seeding particle density, low light intensity, random correlations and large velocity gradients across the integration area. These non-sensible vectors distract from the valid results of PIV as well as causing problems in the calculation of further parameters from the flow being studied, such as vorticity or turbulence intensity. PIV results are normally derived from many pairs of images, so it would be highly impractical to remove all these vectors by hand. Therefore, some kind of sorting algorithm is necessary to find and remove these vectors. The software used in this study[58] uses local median filtering to locate and remove invalid vectors. In this filter, the eight nearest neighbours of a
given vector are sorted into order of increasing magnitude and the median value found. The vector is then removed if its difference in magnitude with the median vector is greater than a critical value. This value is set by the user according to the flow conditions being studied. After the whole vector map has been validated, there will be number of blanks where the vectors have been deleted. These blanks need to be filled if any further analysis is to be carried out on the vector map.
Chapter 5

Presentation of Two-Dimensional Simulation Results

As outlined in section 3.2.2, three-dimensional simulations take a huge amount of computer time to complete. The reduction in computing time offered by two-dimensional simulations allows a wider range of situations to be studied at a reduced computational cost [2, 5]. Two-dimensional simulations also allow the study of a larger number of oscillations than is possible with current computer hardware in three-dimensions. Turbulence is intrinsically a three-dimensional phenomenon and so only limited information can be obtained from such two-dimensional studies. However there are some useful things that can be achieved. There is a well know laminar solution to the Navier-Stokes equation for two-dimensional oscillatory flow, which allows easy measurement of the accuracy of
the LBM model implemented. Several theoretical investigations of the transition to turbulence have used two-dimensional stability theory[63, 64, 65, 66, 67] and the two-dimensional LBM is used to investigate the results of these studies also.

5.0.5 Simulation Parameters

The parameters for all of the simulations in Chapter 5 are detailed in table 5.1. The parameters $M$, $P_{GRAD}$, $\tau$, and the period of oscillation were chosen to give the flow produced by the simulation the desired values of $\alpha$ and $\mathcal{R}_S$. The streamwise grid size was chosen to be an integer multiple of the number of nodes in the Beowulf cluster used to carry out the simulation which lay close to the value of $M$ need to produce the desired flow.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Streamwise grid size ($N$)</td>
<td>128</td>
</tr>
<tr>
<td>Spanwise grid size ($M$)</td>
<td>148</td>
</tr>
<tr>
<td>Period of oscillation</td>
<td>1025826</td>
</tr>
<tr>
<td>Pressure Gradient ($P_{GRAD}$)</td>
<td>$2.11054 \times 10^{-7}$</td>
</tr>
<tr>
<td>Relaxation Time ($\tau$)</td>
<td>0.502053</td>
</tr>
</tbody>
</table>

Table 5.1: Parameters for two-dimensional simulations.
5.1 Comparison of Simulation Results with Analytical Solution of the Navier-Stokes Equation

The D2Q9 lattice-Boltzmann model implemented for this study provides an accurate simulation of the Navier-Stokes equation for two-dimensional oscillatory flow at high values of \( Re_\delta \). The plots in figure 5.1 show the simulated and analytical velocities, in lattice units and the percentage differences. Except where the value of \( u \sim 0 \), the percentage difference between the exact analytical solution and the simulations is less than 1%. Where \( u \sim 0 \), the percentage difference is much greater, most notably in figure 5.1(d). This is caused by the small value of \( u \) in both the simulations and the analytical solution. For figure 5.1(d), a plot showing the absolute value of the difference is shown in figure 5.2. It can be seen from figure 5.2 that the difference between the solution to the Navier-Stokes equation and the results of the lattice-Boltzmann simulation appear quite large in the centre of the channel, \(-0.25 < y < 0.25\). However, the magnitude of the error is only \( \sim 5 \times 10^{-5} \) which is less than 1% of the mean value of \( u = 2.3 \times 10^{-2} \).

The spanwise velocity, \( v \), given by the analytical solution to the Navier-Stokes equation is zero across the whole width of the channel for laminar flow. The lattice-Boltzmann simulations give a value where \( u \sim 10^{-6} \). This arises from the rounding errors due to the limited accuracy with which the computer stores
Figure 5.1: Comparison of streamwise velocity, $u$, profile in a lattice-Boltzmann simulation and the analytical solution to the Navier-Stokes equation at phases $45^\circ$(a), $90^\circ$(b), $135^\circ$(c) and $180^\circ$(d) with $\alpha = 7$ and $Re \sim 710$
Figure 5.2: Comparison of streamwise velocity, $u$, profile in a lattice-Boltzmann simulation and the analytical solution to the Navier-Stokes equation at phase $180^\circ$ with $\alpha = 7$ and $Re_\delta \sim 710$ showing the absolute difference.
the numbers when running the simulation. The small value that the simulation gives to the streamwise velocity does not grow, even if the simulation is run for many oscillations and can be considered to be approximately zero and so in line with the analytical solution to the Navier-Stokes equation. It should be noted that these errors in the value of \( v \) provide a small break in the symmetry of the flow profile in the lattice-Boltzmann simulation, as discussed in section 3.7. This simulation has \( Re \sim 710 \), which is well into the turbulent regime\([5, 2]\) so that if there were a breaking of the symmetry in the flow profile by these rounding errors, then there would be a transition to turbulence. Once this has occurred, the symmetry of the profile would be irrevocably broken and turbulence would be observed in every subsequent oscillation. As this is not the case, it can be assumed that these small fluctuations in the value of \( v \) are not enough to trigger the transition to turbulence. This means that the extra measures discussed in section 3.7 are still necessary and also that the rounding error in the value of \( v \) introduced here does not affect the results of the simulations.

5.2 Development of Turbulence in Two-Dimensional Lattice-Boltzmann simulations

The development of turbulence in a two-dimensional LBM simulation of oscillatory flow has several interesting features. The first of these is the phase at
which the turbulence develops. Figure 5.3 shows how the normalised turbulent intensity, $q^*$, develops with time over the spanwise direction of the channel. The solid line represents the extent of the Stokes boundary layer. The normalised turbulent intensity is defined in equation 2.16. The $x$-component, $\overline{u_x}(y)$ is defined in equation 2.15. The $y$-component of $\overline{u_x}(y)$, $\overline{v_x}(y)$, is zero as there is no mean flow in the spanwise direction. Turbulence begins to develop in this simulation at $\omega t = 2.513$. The value of $q^*$ first shows a significant increase in the Stokes boundary layer, with the disturbance growing and spreading to the centre of the channel with time, peaking where $\omega t = 3.142$. Figure 5.4 shows that this is when the forcing term is at a maximum and the mean streamwise velocity for the laminar profile has just passed the flow reversal. There are also further turbulent bursts at $\omega t = 6.283$, $\omega t = 9.425$ and the beginnings of a burst at $\omega t = 12.566$.

Concentrating on the second and third bursts, there are clear differences in the size and formation of these bursts. As the fourth burst is just beginning as simulation ended, it is not really possible to discuss it in any detail. The second and third bursts are much less intense than the first. Neither develop fully into the centre of the channel. Each of the three burst visible here appear to be most intense in opposing sides of the channel. Although the first burst is evident across the whole with of the channel, it is strongest in the area $-1.0 < y < 0.0$. The second burst is most intense where $0.0 < y < 1.0$, while the third is most intense in the range $-1.0 < y < 0.0$. Both the second and third burst are significantly less intense than the burst which precedes them. The appearance of the most
Figure 5.3: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
Figure 5.4: Plot showing the mean streamwise velocity of a laminar flow and the scaled forcing term with $\alpha = 7.0$, $Re_\delta = 918.6$. The forcing term has been scaled to the mean streamwise velocity for comparison, but it actually varies in the range $-P_{GRAD} \leq F \leq P_{GRAD}$.
intensive section of the burst in the opposite side of the channel to that which proceeded it suggests that the location and intensity of a given burst is dependent on the location and intensity of the burst which preceded it.

**Development of the First Turbulent Burst**

Figure 5.5 shows vector maps of the whole channel at selected phases over the life of the first turbulent burst. Figure 5.5(a) occurs at a time before the turbulence has begun to form. There are some small areas of correlation throughout the flow, most notably in the centre of the channel. In the Stokes boundary layer however, the fluctuating velocities are quite randomly orientated. There are no vortexes in the whole of the flow. This situation has changed by the time of figure 5.5(b). Now, vortexes are clearly beginning to form, with the largest fluctuating velocities located in the Stokes boundary layer next to the channel walls. The clearest vortex is in the range \((0.7 < x < 1.6, -1.0 < y < -0.25)\). Figure 5.5(c) shows that four main vortexes have developed at \(\omega t = 2.945\). The largest fluctuation velocities are still mainly located in the Stokes boundary layer, but there has also been a clear increase in the fluctuating velocities toward the centre of the channel. Figure 5.5(d) shows the fully developed turbulent state. The centres of the four main counter rotating vortexes have moved toward the centre of the channel from those shown in figure 5.5(c). This is the peak of the turbulent activity shown in figure 5.3, with \(\omega t = 3.142\). As the turbulence begins to die away, the
Chapter 5. Presentation of Two-Dimensional Simulation Results

(a)

(b)

(c)

(d)
Figure 5.5: Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_\delta = 918.6$ and $\omega t = 2.356(a)$, $2.749(b)$, $2.945(c)$, $3.142(d)$, $3.587(e)$, $3.927(f)$, $4.320(g)$ and $4.712(h)$. The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 5.3. Therefore, the normalised vector magnitude, is calculated as $|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2}}{|u|}$.
four main vortexes combine to form two counter rotating vortexes in the centre of the channel, as shown in figure 5.5(e). The centres of the two vortexes are in the range $-0.5 < y < -0.2$, rather than the exact centre of the channel, reflecting the fact that the most intense turbulence was in the range $-1.0 < y < 0.0$. The fluctuating velocities near the channel walls have now fallen back to the levels seen in the whole flow in figure 5.5(a). As the turbulence develops further, the reduction in size of the fluctuating velocities in the Stokes Layer spreads further into the centre of the channel, depicted in figure 5.5(f). The fluctuating velocities have also reduced in magnitude in the centre of the two main vortexes. Both of these trends continue as the turbulence dies away in figure 5.5(g). Here, the two main vortexes have broken down, although it is still possible to see a 'ghost' of the vortexes in the range $-0.5 < y < 0.0$. Figure 5.5(h) show the situation at $\omega t = 4.712$. It can be said that the turbulence has died away completely, with the situation broadly similar to that in figure 5.5(a) rather than in 5.5(d-f).

The phase and manner with which turbulence develops in this simulation is comparable with the work by Akhavan et al[1] which is the only previous major attempt to simulate the development of turbulence in a bounded oscillatory flow.

**Development of the Second Turbulent Burst**

The second turbulent burst is less intense than the first, as shown in figure 5.3. Figure 5.6 shows vector maps over the whole channel at the equivalent
Chapter 5. Presentation of Two-Dimensional Simulation Results

(a) Channel Length Normalised to Width

(b) Channel Length Normalised to Width

(c) Channel Length Normalised to Width

(d) Channel Length Normalised to Width
Figure 5.6: Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_\delta = 918.6$ and $\omega t = 5.498(a)$, 5.890(b), 6.087(c), 6.283(d), 6.676(e), 7.069(f), 7.461(g) and 7.854(h). The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 5.3. Therefore, the normalised vector magnitude, is calculated as $|u'|^* = \frac{|u'|}{|u|} = \sqrt{u'^2 + v'^2}$. 
phases in the cycle shown in figure 5.5 for the first burst. This allows for a
direct comparison over the whole turbulent burst. Figure 5.6(a) shows that the
fluctuation velocities are randomly directed over the whole channel, which appears
very similar to the situation in figure 5.5(a). At this phase in the first turbulent
burst, while the fluctuating velocities are of a similar magnitude, there are some
areas of correlation in the centre of the channel. These are not present at all in
figure 5.6(a). Figure 5.6(b) shows evidence of turbulence developing in the Stokes
boundary layer at $-1.0 < y < -0.8$. The situation here is comparable to that in
the range $0.8 < y < 1.0$ in figure 5.5(b), but the development is not as intense as
in the range $-1.0 < y < -0.8$ in figure 5.5(b). This contrast is further highlighted
at the height of the turbulent intensity. In figure 5.6(c-e), the vortexes formed
are much less distinct and form later and die away quicker than those in figure
5.5(c-e). While the maximum size of the fluctuating velocities are comparable in
both the first and second bursts, those in the second burst occur at a later time
and over a smaller area. The vortexes formed in the second turbulent burst are
not as regular as those in the first burst. Figure 5.6(e-f) show that rather than
merging into two vortexes in the centre of the channel, a number of small vortexes
are formed which then appear to die away more quickly. While the magnitude of
the fluctuating velocities have died away when $\omega t = 7.069$, there are still areas of
correlation and it is only when $\omega t = 7.854$, shown in figure 5.6(h), that the burst
has truly died away.
Development of the Third Turbulent Burst

The development of the third turbulent burst is different again from the proceeding two. Turbulence begins to develop at a later phase in the cycle. The situation in figure 5.7(b), rather than showing the beginning of turbulence development shown in figures 5.5(b) and 5.6(b), is very similar to figure 5.7(a). At this phase, there are only small areas of fluctuating velocity correlation mostly located in the Stokes boundary layer. Figures 5.7(c-e) show that the burst develops in a similar way to that of the second turbulent burst, but with the largest fluctuating velocities in the opposite side of the channel. However, the vortexes formed in the third turbulent burst bare more similarities to those in the second turbulent burst than the first. Figure 5.7(d) indicates that coherent vortexes have only formed in the range \(-1.0 < y < 0.0\), with little evidence of vortexes in the other half of the channel. Figure 5.7(e) shows that rather than merging into two counter-rotating vortexes in the centre of the channel, as in figure 5.5(e), several vortexes of different size and intensity form. A comparison of figures 5.7(f-g) with figures 5.5(f-g) shows that the vortexes, while forming at a similar rate to those in the first turbulent burst die away in a shorter time, with virtually no trace of them remaining when \(\omega t = 10.603\).
Chapter 5. Presentation of Two-Dimensional Simulation Results

(a) 

(b) 

(c) 

(d)
Figure 5.7: Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_\delta = 918.6$ and $\omega t = 8.639$ (a), 9.032(b), 9.228(c), 9.425(d), 9.817(e), 10.210(f), 10.603(g) and 10.996(h). The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 5.3. Therefore, the normalised vector magnitude, is calculated as $|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2}}{|u|}$. 

93
Figure 5.8: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
Figure 5.9: Contour plot of the normalised turbulent intensity, \( q^* \) across the width of the channel width \( \omega t \) with \( \alpha = 7.0, Re_\delta = 918.6 \). The horizontal lines at \( \pm 0.80 \) indicate the width of the Stokes boundary layer.
Further Observations

The simulation results shown in figure 5.3 indicate that the turbulent intensities are greater when the main streamwise velocity is in the positive direction. Figure 5.8 shows that this is generally not the case. This simulation uses identical parameters to that shown in figure 5.3, except that the first turbulent burst has been delayed by a factor of $\omega t = \pi$. This has produced the same pattern of intense, non-intense, intense turbulent bursts as seen in figure 5.3, but with a phase lag of $\pi$. This shows that the sequence of intense and non-intense turbulent burst is independent of the flow direction. It suggests that the change in the level of intensity in the turbulent bursts is related to the intensity of the burst which precedes it. The second turbulent burst has a low intensity and is preceded by a high intensity burst. The third turbulent burst is high intensity and is preceded by a low intensity burst. The first turbulent burst is essentially preceded by laminar flow, which is equivalent to a turbulent burst of zero intensity. Looking at all the bursts in the two simulations, it seems that the intensity of a burst is highly influenced by the intensity of the previous bursts. This leads to an oscillation in the intensity of the turbulent bursts, as a high intensity burst is followed by a low intensity one. Figure 5.9 shows a continuation of the simulation in figure 5.3. The turbulent bursts continue in a pattern which seems to confirm that the size and also location of a turbulent burst is highly influenced by the previous burst. The third turbulent burst in figure 5.3 is slightly more intense in the lower
half of the channel. This then feeds into the fourth burst (the first in figure 5.9, at $\omega t = 12.566$) being most intense in the top half of the channel. This pattern continues over the remaining turbulent bursts. In the top half of the channel, the burst where $\omega t = 18.850$ is more intensive than the ones in the bottom half of the channel at $\omega t = 15.708$ and $\omega t = 21.991$. A further possibility is the existence of two solutions, with only one shown in these simulation results. As the flow develops past the first and second turbulent bursts, clear pattern emerges where a turbulent burst occurs first in one half of the channel, with little or no turbulent activity in the other half, and then the reverse situation occurs at the next turbulent point in the cycle. It is possible that the solution calculated by the two-dimensional lattice-Boltzmann simulation is only one of two possible solutions, with the second solution being the mirror image of alternating turbulent bursts along the line $y = 0$. The true flow in this case would then be a combination of these two solutions.

5.2.1 Reynolds Stress and Turbulent Energy Cascade

Figure 5.10 shows a plot of the normalised Reynolds stress over the width of the channel. This plot shows that there is very little Reynolds stress in a two-dimensional development of turbulence. The value of $\varphi^*$ is only significant at the height of the turbulent bursts. The sign of $\varphi^*$ signifies the general movement of the fluctuating velocities in the flow. As outlined in section 2.3.3, the sign of $\varphi^*$
gives an indication of what is happening in the flow. When $\varphi^* > 0$, as in the range $-0.8 < y < 0.3$ at $\omega t = 3.142$ in figure 5.10, this means that the values of $u'$ and $v'$ generally both have the same sign. Figure 5.4 shows that the overall velocity is negative at this point. Therefore a positive Reynolds Stress at this point, $\overline{u'v'} < 0$, indicates that a particle moving from the Stokes boundary layer into the main flow, $v' > 0$, will have slower streamwise velocity, $u' < 0$, than the surrounding particles and vice-versa for particles moving onto the Stokes boundary layer. The relative movement of particles is reversed where $\varphi^* < 0$ in the range $-0.5 < y < 0.8$, as the movements parallel to the spanwise direction into and out of the Stokes boundary layer have the opposite sign. For the second turbulent burst, this pattern is repeated with the opposite signs for the Reynolds stress, as the overall velocity is positive at this point, although the area where it would be expected that $\varphi^* < 0$, somewhere in the range $-1 < y < -0.2$, is not visible. This is due to the low magnitude of the turbulent intensity in this area of the flow. In the third turbulent burst, the same pattern as in the first turbulent burst is seen, which is to be expected, with the overall velocity being in the same direction. Again, as in the second turbulent burst, the magnitude of the Reynolds stress is low where the turbulent intensity is low.

The energy spectra of the flow at different stages in the development of the first turbulent burst is shown in figure 5.11. There are several interesting features. The most obvious is the appearance of the Kolmogorov $-\frac{5}{3}$ law in the spectra for $\omega t = 3.142$ and $\omega t = 3.534$. Although these do not match up exactly with
Figure 5.10: Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
Figure 5.11: A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 7.0$, $Re_\delta = 918.6$. 
Chapter 5. Presentation of Two-Dimensional Simulation Results

the $-\frac{5}{3}$ line plotted on the graph, there is a clear gradient of $\sim -\frac{5}{3}$ in the range $0.1 < k_d < 0.2$. As the burst begins to decay, at $\omega t = 3.927$ and $\omega t = 4.32$, there is still a large amount of energy for low values of $k_d$. In the range $0.1 < k_d < 0.2$ where the Kolmogorov $-\frac{5}{3}$ law was most clearly shown at $\omega t = 3.142$ and $\omega t = 3.534$, there is a minimum in the energy spectra. Another feature of all the spectra over this range of the simulation is the increase in the energy at higher values of $k_d$. This is in direct contrast to established behaviour where the energy decrease, first at a rate of $-\frac{5}{3}$, then faster as $k_d$ increases[23, 68]. Both of these features indicate that there is a problem with the simulations. The increase in energy at higher values of $k_d$ seems to suggest that the simulation is not resolving the smaller scales to a great enough accuracy.

5.3 Conclusions

The use of the two-dimensional simulations has provided a good starting point for the larger and much more computationally expensive three-dimensional simulation. By indicating the phase of the oscillation at which turbulence begins to develop, it has provided information on the optimum phase to begin the larger simulations to extract the largest amount of information from a practical runtime. The two-dimensional simulations have also given an indication of the possible problems with the three-dimensional simulations. At this stage, it was hoped that the three-dimensional simulations would give a good direct nu-
numerical simulation of turbulence in oscillatory channel flow. The energy spectra of the two-dimensional simulations indicate that this will possibly not be the case. They show that for two-dimensional simulations, a much higher lattice resolution is needed to provide a complete picture of turbulent flow over all the important length scales. This will not be possible with the three-dimensional simulations on the available computing resources.
Chapter 6

Presentation of

Three-Dimensional Channel

Simulation Results

6.1 A Three-Dimensional Channel Model

This section concerns three-dimensional lattice-Boltzmann simulations in an infinite channel, as illustrated in figure 6.1. The streamwise and spanwise directions are still labelled as $x$ and $y$ respectively. The $z$-direction is known as the cross-stream direction. Periodic boundary conditions are implemented at the extremities of the $z$-direction to create an infinite channel. As in the two-
Chapter 6. Presentation of Three-Dimensional Channel Simulation Results

Figure 6.1: Diagram of the computational domain for the three-dimensional channel simulations.

During three-dimensional simulations, there is a periodic boundary condition at edges of the streamwise direction. The velocity profile of the streamwise flow in this channel will be the same as that in a two-dimensional channel, due to the periodic boundary conditions in the z-direction, and so can be computed using equation 2.10. For the spanwise and cross-stream directions, \( v = w = 0 \) in the laminar regime.

As the simulation times for a three-dimensional LBM are \( \sim 300 \) times that of a two-dimensional simulation, section 3.2.2, it is only possible to run a simulation for one or two oscillations. Although this flow has the same streamwise velocity profile as the two-dimensional simulations seen in Chapter 5, it allows fully three-dimensional turbulent structures to develop.

6.1.1 Simulation Parameters

The parameters for all of the simulations in Chapter 6 are detailed in table 6.1. These parameters are the same as those used in the two-dimensional simulations.
### Chapter 6. Presentation of Three-Dimensional Channel Simulation Results

#### Table 6.1: Parameters for three-dimensional channel simulation.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Streamwise grid size ((N))</td>
<td>128</td>
</tr>
<tr>
<td>Spanwise grid size ((M))</td>
<td>148</td>
</tr>
<tr>
<td>Cross-Stream grid size ((L))</td>
<td>128</td>
</tr>
<tr>
<td>Period of oscillation</td>
<td>1025826</td>
</tr>
<tr>
<td>Pressure Gradient ((PGRAD))</td>
<td>(2.11054 \times 10^{-7})</td>
</tr>
<tr>
<td>Relaxation Time ((\tau))</td>
<td>0.502053</td>
</tr>
</tbody>
</table>

There are marked differences between the development of turbulence in two-dimensions and in the three-dimensional channel LBM. Figure 6.2 shows the development of \(q^*\) across the spanwise direction of the three-dimensional channel. Note that there is a slight difference in the value of \(Re_\delta\) compared to the two-dimensional simulations. \(Re_\delta\) is calculated from the maximum flow recorded in the simulation and so the its higher value in this simulation indicates that the maximum velocity reached in this simulations is higher than in the two dimensional simulations. For the three-dimensional channel simulation, the turbulent intensity is defined in a slightly different way to account for the additional dimension:

\[
q^* = q \frac{|u|}{\overline{|u|}} = \frac{\sqrt{u'^2 + v'^2 + w'^2}}{|u|} \tag{6.1}
\]
where \( w' \) is the \( z \)-component of \( u' \), as defined in equation 2.14. Definitions for \(|\overline{u}|, \overline{u'^{2}}\) and \( v'^{2} \) can be found in section 5.2. The \( z \)-component of \( \overline{u_{x}}(y) \) (equation 2.14), \( \overline{u_{x}}(y) = 0 \) as there is no mean flow in the cross-stream direction, as is the case for \( \overline{v_{x}}(y) \) in the spanwise direction. For three-dimensional simulations,

\[
\overline{u_{x}}(y) = \frac{\sum_{x=1}^{N} \sum_{z=1}^{L} u(x, y, z)}{NL}
\]  

(6.2)

Also, \( u'^{2} \) and \( v'^{2} \), along with \( w'^{2} \) are the \( x \)-, \( y \)-and \( z \)-components of the three-dimensional version of \( u'^{2} \), which is defined as:

\[
\overline{u'^{2}} = \frac{\sum_{x=1}^{N} \sum_{z=1}^{L} u'^{2}(x, y, z)}{NL}
\]  

(6.3)

Figure 6.3 shows the same phase of the two-dimensional simulation as figure 5.3, with the contour values of \( q^{*} \) scaled to match those in figure 6.2. The most obvious difference is that the magnitude of \( q^{*} \) is much greater in the three-dimensional channel simulation. At its most intense, it is approximately twice as large as in the two-dimensional simulation. There is also a great difference in the phase at which the turbulence develops. In the two-dimensional simulation, the first turbulent burst develops to its peak intensity at \( \omega t = 3.142 \). In the three-dimensional channel simulation, the development of the first turbulent burst is more complex. There is an initial increase in the turbulent intensity peaking at \( \omega t = 2.670 \) with approximately the same magnitude as the first burst in the two-dimensional simulation. The turbulent intensity then dies away in the centre of the channel at
Figure 6.2: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 987.47$. The horizontal lines at $\pm0.80$ indicate the width of the Stokes boundary layer.
Chapter 6. Presentation of Three-Dimensional Channel Simulation Results

Figure 6.3: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
$\omega t = 3.142$, while continuing in the Stokes boundary layer. Indeed, as $\omega t = 3.142$, the turbulent intensity has begun to increase again in the Stokes boundary layer. The higher intensity in the boundary layer begins to move into the main flow at $\omega t = 3.770$. The turbulent intensity then increases to a maximum at $\omega t = 4.241$. Unlike the two-dimensional simulation, the magnitude of $q^*$ peaks in the ranges $0.85 > y > 0.75$ and $-0.75 > y > -0.85$ instead of the peak occurring in the centre of the channel.

These are the two distinct stages in the first turbulent burst in the three-dimensional channel simulation. The first of these, the primary disturbance, peaks as $\omega t = 2.670$. The second or secondary disturbance peaks as $\omega t = 3.770$. The primary disturbance bears similarities to the situation in the two-dimensional simulation. The phase at which the burst starts are similar for both simulations. The development of the turbulence also follows a similar pattern, with the initial disturbances generated in the Stokes layer followed by a movement to the centre of the channel. There are differences, however. The turbulent intensity in the Stokes boundary layer starts to increase much earlier and is constantly at a higher intensity in the three-dimensional channel simulation. The primary disturbance also dies away much more quickly in the centre of the channel than the turbulent burst in the two-dimensional simulation. The greatest contrast occurs at $\omega t = 3.299$. At this point in the two-dimensional simulation, the turbulent intensity is still close to its maximum value, while in the three-dimensional channel simulation, the intensity in the centre of the channel has dropped so that the
value of $q^*$ is close to zero. The secondary disturbance in the three-dimensional channel simulation does not appear in the two-dimensional simulation at all. At this point in the cycle, the turbulent intensity of the two-dimensional simulation is dying away to zero, while in the three-dimensional channel simulation it is building to its maximum value.

**Dimensionality of the Primary and Secondary Disturbances**

It has been suggested[69, 33] that there are two distinct phase to the generation of turbulence, known as the Primary and Secondary Instabilities. The primary instability is thought to be a largely two-dimensional phenomenon, even in full three-dimensional flows. It has been shown[70, 71, 72] that the breakdown of these two-dimensional vortexes catalyses the transition to full three-dimensional turbulence seen in the Secondary Instability. Figure 6.4 shows the normalised components of the fluctuation velocity, $u'$:

\[
\begin{align*}
    u'^* &= \frac{\sqrt{u'^2}}{q}, \\
    v'^* &= \frac{\sqrt{v'^2}}{q}, \\
    w'^* &= \frac{\sqrt{w'^2}}{q}
\end{align*}
\]  

(6.4)

This means that the size of the velocity component at any given point in the flow is only relative to the other two components at that point. The most interesting feature of figure 6.4 occurs in the range $2.199 < \omega t < 2.513$. Figure 6.4(c) indicates that the $z$-component of the velocity, figure 6.4(c), falls to $\sim 0.1$ in the centre of the channel and is significantly smaller than the $x$- and $y$-components
Chapter 6. Presentation of Three-Dimensional Channel Simulation Results

Figure 6.4: Contour plot of the normalised components of the velocity, $u^*$ (a), $v^*$ (b) and $w^*$ (c) across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 987.47$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
in the same phase range, figure 6.4(a+b). Before this time, $1.571 < \omega t < 2.120$, the velocity is fairly evenly distributed between the three velocity components in the centre of the channel. The situation in the Stokes boundary layer is different. Next to the channel wall, there is a slight dominance of the $x$-component at $\omega t = 1.571$. This then moves to the edge of the Stokes boundary layer at $\omega t = 2.513$. After a similar streak of the $y$-component at slightly later time, the $x$-component begins to dominate again and continues to do so until then the end of the Secondary Instability ($\omega t = 4.712$). The fall of the value of the $z$-component just before the primary instability indicates that vortexes formed in this phase of the flow are two-dimensional, rotating in the $x - y$ plane. During the secondary instability, $2.827 < \omega t < 3.770$, the $y$-component dominates in the centre of the channel, while the $x$-component dominates in the Stokes boundary layer. This is illustrated in figure 6.5, which shows a full three dimensional vector map of the fluctuation velocity over the whole flow. The three components of each vector in figure 6.5 are calculated as an average of a $6 \times 6 \times 6$ cube of points from the simulation lattice. Figure 6.5(a) occurs at $\omega t = 2.553$. This is just before the peak of the primary instability. This plot is taken at the end of the period where the $z$-component of the fluctuation velocity, $w^* \sim 0.1$. In the top half of the channel, $0 < y < 1$, $w^* \sim 0.1$. This can be seen in figure 6.5(a) as the vectors are largely in the $x - y$ plane. In the bottom half of the channel shown in figure 6.5(a), the velocity components are more evenly distributed at $\omega t = 2.553$, shown in figure 6.4, and this is reflected in the orientation of the vectors. Figure
Chapter 6. Presentation of Three-Dimensional Channel Simulation Results
Chapter 6. Presentation of Three-Dimensional Channel Simulation Results
Chapter 6. Presentation of Three-Dimensional Channel Simulation Results

(d)
Figure 6.5: Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_d = 987.47$ and $\omega t = 2.553(a)$, $3.142(b)$, $3.731(c)$, $4.320(d)$ and $4.909(e)$. The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 6.2. Therefore, the normalised vector magnitude, is calculated as $|u'|^* = \frac{|u'|}{|u|} = \frac{\sqrt{u'^2 + v'^2 + w'^2}}{|u|}$. The size of the vectors is scaled by their relative magnitude.
6.5(b) shows the vectors as $\omega t = 3.142$. Here, the primary instability has all but died away. Figure 6.5(c) shows the situation as the secondary instability has begun to develop at $\omega t = 3.731$. The largest vectors are situated in the Stokes boundary layer, which is to be expected. The turbulence has not yet developed in the centre of the channel, shown in figure 6.2, and consequently the fluctuating velocity vectors here are very much smaller than those in the boundary layer. Figure 6.5(d) shows a vector map of the fluctuating velocities at the peak of the secondary instability. All of the visible vortexes have no obvious bias to any particular plane and are clearly three-dimensional in nature, as shown in figure 6.4. Figure 6.5(e) shows the vector map as the secondary instability is dying away at time $\omega t = 4.909$.

6.2.1 Reynolds Stress and Turbulent Energy Cascade

Figure 6.6 shows a contour plot of the mean Reynolds stress over the width of the channel as the turbulence develops. A comparison with the development of $q^*$, shown in figure 6.2, gives a further indication of how turbulence develops. Although there is no visible Reynolds stress during the primary instability, where $2.827 < \omega t < 3.770$, this is due to the scaling of the contours in the plot. The small area of significant Reynolds stress at $\omega t = 2.670$, in the range $-0.75 < y < -0.6$, show that there is still Reynolds stress in the primary instability and it has the same sign as the first turbulent burst shown in figure 5.10. The sign of the
Reynolds stress is then opposite for the secondary instability. A comparison of the second turbulent burst in the two-dimensional simulation, figure 5.10, and the second burst in figure 6.6 shows that the second bursts in the two different simulations have opposing signs. Taking the half of the channel where \(-1 < y < -0\) in the three-dimensional channel simulation, the change in sign of the Reynolds stress occurs as the mean velocity of the flow changes from accelerating, in the primary instability to decelerating in the secondary instability. Figure 6.7 shows the energy spectra at different phases over the development of the first turbulent burst. While none of the spectra exactly match the Kolmogorov \(-\frac{5}{3}\) law, once the turbulence develops, all the spectra have a clear gradient of \(\sim -\frac{5}{3}\). The spectra becomes slightly less clear as the turbulent intensity lessens in the centre of the channel between the primary and secondary instability, with \(\omega t = 3.534\), but then the Kolmogorov \(-\frac{5}{3}\) law becomes even more apparent during the secondary instability. However, as in the two-dimensional simulations, figure 5.11, the higher values of \(k_d\) increase in energy. This again points to a lack of resolution at small scales, although it is interesting to note that this problem is much less pronounced for the fully three-dimensional secondary instability \((3.927 < \omega t < 4.712)\) in the three-dimensional channel simulation than it is for either the two-dimensional simulations or the primary instability in the three-dimensional channel simulation \((2.749 < \omega t < 3.142)\), where it has been suggested that the turbulence is largely a two-dimensional phenomena.
Figure 6.6: Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 987.47$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
Figure 6.7: A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 7.0$, $Re_\delta = 987.47$. 
6.3 Conclusions

The three-dimensional channel simulations provide an interesting stepping stone from the two-dimensional simulations to the three-dimensional duct simulations, showing a two stage transition to turbulence. The primary instability occurs at broadly the same phase as that in the two-dimensional simulations, with the secondary instability occurring at the expected phase for full three-dimensional turbulence. Closer inspection shows that the primary instability is largely a two-dimensional phenomena, with the vortexes increasing in size over the primary instability and then reducing in intensity and distorting in to full three-dimensional structures at the secondary instability develops. The energy spectra shows that there is a lack of resolution at small scales, which could be distorting the picture. However the spectra follow the Kolmogorov $-\frac{5}{3}$ law much better than those calculated from the two-dimensional simulations and it seems unlikely that the lack of resolution has had a significant effect on the development of turbulence.
Chapter 7

Presentation of

Three-Dimensional Duct

Simulation Results

7.1 A Three-Dimensional Duct Model

This chapter concerns a three-dimensional lattice-Boltzmann simulation in a square duct. A schematic of the simulation is shown in figure 7.1. The $x$-direction is the streamwise direction and contains periodic boundary conditions to create an infinitely long duct. The $y$- and $z$-directions, the spanwise and cross-stream directions, are implemented with a bounce-back boundary condition to create a
Figure 7.1: Diagram of the computational domain for the three-dimensional duct simulations.

square duct. The velocity profile used to initialise these simulations was computed from the analytical solution of the Navier-Stokes equation given in section 2.2.2. This gives velocities for the streamwise direction. The velocities in the spanwise and cross-stream direction were set to zero. This is only an approximation for flows in a square duct as there are non-zero velocities in both the streamwise and cross-stream directions for laminar flow in a square duct. However, these are small and there are no analytical solutions available to compute them, so they have been approximated to zero in the initialisation of these simulations. As the simulations begins, this will be corrected within ~ 100000 timesteps (approximately $\frac{1}{10}$ of a period).
7.1.1 Simulation Parameters

The parameters for all of the simulations in Chapter 7 are detailed in table 7.1. These are the same as in the two-dimensional and three-dimensional channel simulations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Streamwise grid size ((N))</td>
<td>128</td>
</tr>
<tr>
<td>Spanwise grid size ((M))</td>
<td>148</td>
</tr>
<tr>
<td>Cross-Stream grid size ((L))</td>
<td>148</td>
</tr>
<tr>
<td>Period of oscillation</td>
<td>1025826</td>
</tr>
<tr>
<td>Pressure Gradient ((P_{GRAD}))</td>
<td>(2.11054 \times 10^{-7})</td>
</tr>
<tr>
<td>Relaxation Time ((\tau))</td>
<td>0.502053</td>
</tr>
</tbody>
</table>

Table 7.1: Parameters for three-dimensional duct simulation.

7.2 Development of Turbulence in three-dimensional duct Lattice-Boltzmann simulations

The development of turbulence in the three-dimensional duct simulation is very similar to that in the three-dimensional channel simulation. This is not surprising, due to the three-dimensional nature of these simulations. There are, however, some significant differences. Figure 7.2 shows the development of the turbulent intensity, \(q^*\), with the contour values of \(q^*\) scaled to match those in figure 6.2 to allow a direct comparison. Note that the value of \(Re_s\) is lower here than in the three-dimensional channel simulation. This indicated that the ad-
dition of a solid boundary in the cross-stream direction in this simulation has restricted the maximum velocity generated by the flow compared to the periodic boundary condition in the three-dimensional channel simulation. The most obvious difference is that the overall intensity of the turbulence is significantly less in the three-dimensional duct simulation, with the maximum value of $q^* \sim 10\%$ smaller than in the three-dimensional channel simulation. There is also a difference in the phase of the development of turbulence. The primary instability in the first turbulent burst occurs as $\omega t \sim 2.827$. This matches much more closely the phase of turbulence development in the two-dimensional simulations, shown with the contour values of $q^*$ rescaled, in figure 6.3. The second turbulent burst in the two-dimensional case also closely matched that phase of the second turbulent burst in the three-dimensional duct simulation. Again the difference between the first and second turbulent burst indicates that the formation of a turbulent burst is highly influenced by the burst which preceded it. It is likely that the lack of visible primary and secondary instabilities in the second turbulent burst is caused by the presence of a burst in the preceding cycle. The value of $q^*$ in the centre of the channel in the three-dimensional duct simulation shows a significant difference to that in the three-dimensional channel and two-dimensional simulations. In the three-dimensional duct simulation, the value of $q^*$ remains significantly higher in the centre of the channel between the primary and secondary instabilities in the first turbulent burst and also remains significantly higher across the whole width of the channel between the first and second turbulent bursts.
Figure 7.2: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 884.77$. The horizontal lines at ±0.80 indicate the width of the Stokes boundary layer.
This can be accounted for by the difference in the flow configuration between the three-dimensional duct and the other simulations. Both the two-dimensional and three-dimensional channel simulations are channel flow. However, the three-dimensional duct simulation is duct flow. This means that there is a boundary wall along all the extremities of both the y- and z-directions, as indicated in figure 7.1. As a result, the higher levels of $q^*$ seen in the boundaries where $y < -0.80$ and $y > 0.80$ will also occur where $z < -0.80$ and $z > 0.80$. As the calculation of $q^*$ averages over the $z$-direction, this means that the value of $q^*$ will increase over the whole of the channel width (y-direction) as the turbulent fluctuations will increase where $z < -0.80$ and $z > 0.80$. The increase in the value of $z < -0.80$ in the centre of the channel in figure 7.2 also indicates that the turbulent intensity in the Stokes boundary layer is greater toward the centre of a channel wall, than toward the edge, in the corners of the duct.

**Dimensionality of the Primary and Secondary Disturbances**

Figure 7.3 shows the relative sizes of the three components of the fluctuating velocity over the $y$-direction. Comparing this to the same analysis for the three-dimensional channel simulation, there are several key differences between the two different simulations. One of the key features of the three-dimensional channel simulation was the drop to $\sim 0.1$ of $w'^*$ in the range $2.199 < \omega t < 2.513$. This is as the primary instability develops in the first turbulent burst. In the three-
Figure 7.3: Contour plot of the normalised components of the velocity, $u^*$ (a), $v^*$ (b) and $w^*$ (c) across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\theta = 884.77$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
dimensional duct simulation, there is no corresponding drop in the value of \( w' \) as the primary instability develops in the range \( 2.513 < \omega t < 2.827 \). At this point in the cycle, the value of \( w' \) does decrease, but this also happens for the value of \( u' \), as the fluctuating velocity in the streamwise direction grows with \( u' \approx 0.7 \).

The value of \( u'' \) increases again as the secondary instability develops in the first turbulent burst and also as the second turbulent burst develops. The peak values of \( u'' \) end at \( \omega t \approx 4.084 \) for the secondary instability in the first turbulent burst and \( \omega t \approx 6.283 \) for the second turbulent burst. In both cases, this corresponds with the start of a significant increase in the value of the turbulent intensity, \( q^* > 0.152 \), across the whole width of the channel. This indicates that it is an increase in relative magnitude of the streamwise component of the fluctuating velocity which causes the transition to turbulence. By combining the information in figures 7.2 and 7.3, the following qualitative picture of the development of turbulence can be obtain. Taking the development of the secondary instability as an example and begin at time \( \omega t = 3.142 \); as the turbulent intensity, \( q^* \), begins to increase in the stokes boundary layer, the streamwise component of the fluctuating velocity, \( u'' \), becomes dominant in the Stokes boundary layer. When \( \omega t = 3.456 \), the magnitude of \( u'' \) has begun to increase across the whole width of the channel. The value of \( q^* \) has also begun to increase in the boundary layer. As \( \omega t = 3.770 \), the turbulent intensity has begun to increase across the whole width of the channel, with still greater increases in the boundary layer. \( u'' \) is now the dominant component of the fluctuating velocity across the whole width of the channel.
Chapter 7. Presentation of Three-Dimensional Duct Simulation Results

channel. At $\omega t = 4.084$, $q^*$ increases across the whole width of the channel, as the major turbulent activity begins. At the same time, there is an increase in the values of $v^*$ and $w^*$, although $u^*$ is still noticeably larger than either of the other two components. As the increase in $q^*$ corresponds with the increases in $v^*$ and $w^*$, it is likely that the increase in turbulent intensity has been channelled into the spanwise and cross-stream components of the fluctuating velocities, which causes the distortion and twisting of the turbulent vortexes which can be seen in the vector maps in figure 7.4. The fluctuating velocity vectors are displayed at the height of the primary instability in figure 7.4(a), where $\omega t = 2.945$. Comparing this to 6.5(a), the picture is much more confused. This is because in the three-dimensional duct simulation, the additional walls at the extremities of the $z$-direction give the laminar flow an extra axis of symmetry. Instead of essentially two-dimensional vortexes rotating in the $x$-$y$ plane, a more complex situation develops with vortexes rotating in most planes. It is interesting to note that, as shown in figure 7.3, the $x$-component of the fluctuating velocity dominates. This implies that, although the vortexes are rotating in varying planes, most vortexes must be rotating in a plane containing the $x$-direction. Figure 7.4(b) shows the fluctuating velocity vectors after the primary instability has died away. Although most of the vectors are small at this time, it is clear that the largest vectors are in the Stokes boundary layer, as indicated in figure 7.2. There are also larger vectors in the centre of the $y$-direction at the $z = 1.0$ boundary. While these are smaller than those in the centre of the $z$-direction on the $y = 1.0$
channel. At $\omega t = 4.084$, $q^*$ increases across the whole width of the channel, as
the major turbulent activity begins. At the same time, there is an increase in the
values of $v^*$ and $w^*$, although $u^*$ is still noticeably larger than either of the other
two components. As the increase in $q^*$ corresponds with the increases in $v^*$ and
$w^*$, it is likely that the increase in turbulent intensity has been channelled into the
spanwise and cross-stream components of the fluctuating velocities, which causes
the distortion and twisting of the turbulent vortexes which can be seen in the
vector maps in figure 7.4. The fluctuating velocity vectors are displayed at the
height of the primary instability in figure 7.4(a), where $\omega t = 2.945$. Comparing
this to 6.5(a), the picture is much more confused. This is because in the three-
dimensional duct simulation, the additional walls at the extremities of the $z$-
direction give the laminar flow an extra axis of symmetry. Instead of essentially
two-dimensional vortexes rotating in the $x$-$y$ plane, a more complex situation
develops with vortexes rotating in most planes. It is interesting to note that,
as shown in figure 7.3, the $x$-component of the fluctuating velocity dominates.
This implies that, although the vortexes are rotating in varying planes, most
vortexes must be rotating in a plane containing the $x$-direction. Figure 7.4(b)
shows the fluctuating velocity vectors after the primary instability has died away.
Although most of the vectors are small at this time, it is clear that the largest
vectors are in the Stokes boundary layer, as indicated in figure 7.2. There are
also larger vectors in the centre of the $y$-direction at the $z = 1.0$ boundary. While
these are smaller than those in the centre of the $z$-direction on the $y = 1.0
Chapter 7. Presentation of Three-Dimensional Duct Simulation Results

(a)
Chapter 7. Presentation of Three-Dimensional Duct Simulation Results
Chapter 7. Presentation of Three-Dimensional Duct Simulation Results

(d)
Figure 7.4: Vector maps of the fluctuating velocities with $\alpha = 7$, $Re_3 = 884.77$ and $\omega t = 2.945(a)$, 3.534(b), 4.123(c), 4.712(d) and 5.301(e). The normalised magnitude of the vectors, $|u'|^*$, is indicated by their colour, which is scaled so as to be comparable to $q^*$ in figure 6.2. Therefore, the normalised vector magnitude, is calculated as $|u'|^* = \frac{u'}{|u|} = \frac{\sqrt{u'^2 + v'^2 + w'^2}}{|u|}$. The size of the vectors is scaled by their relative magnitude.
boundary, they do account for the increase in the value of $q^*$ seen at the centre of the channel in figure 7.2. The secondary instability develops much more evenly across the spanwise direction in the three-dimensional duct simulation, rather than developing in the Stokes boundary layer and then moving into the centre of the channel in the three-dimensional channel simulation. This is reflected in figure 7.4(c). It is difficult to discern individual vortexes in this plot, but the size of the fluctuating velocities through the whole duct is in contrast with figure 6.5(c) which is taken at a similar phase in the turbulent cycle. In the three-dimensional channel simulation, the largest fluctuation velocities are close to the boundaries at $y = \pm1.0$ with no significant fluctuating velocities in the centre of the channel. The quicker development of the turbulent state in the centre of the duct in the three-dimensional duct simulation can be attributed to the addition of boundaries at $z = \pm1.0$. As turbulence is initially generated close to the boundary, the increase in the boundary surface area has decreased the time taken for the whole duct to reach the turbulent state. The additional boundaries also appear to restrict the maximum size of the turbulent intensity. Figures 7.4(d) and 6.5(d) are taken at roughly the same phase in the turbulent development. In the three-dimensional duct simulation, the turbulent intensity has already peaked and the begun to die away, when compared with the situation in figure 7.4. In the three-dimensional channel, the turbulent intensity has just reached its peak. The vortexes seem to be more evenly distributed between all planes in this more developed turbulent state. There is however a slight
dominance of the $x$-component, shown in figure 7.3. As the turbulence dies away in the duct simulation, it is interesting to note that the value of $q^*$ reduces in the Stokes boundary layer before the centre of the duct, rather than persisting in the boundary layer as in the three-dimensional channel. In this respect it bears more similarity to the two-dimensional simulations, where $q^*$ dies in the boundary layer significantly earlier than in the centre of the channel. This is illustrated in figure 7.4(e).

7.2.1 Reynolds Stress and Turbulent Energy Cascade

The Reynolds Stress in the three-dimensional duct simulation is similar to that in the three-dimensional channel simulation. The Reynolds Stress follows the same pattern as that in the three-dimensional channel simulation, indicating that the development of turbulence in both cases follows the same pattern, with the sign of the Reynolds Stress changing between the primary and secondary instabilities in the first turbulent burst and then changing again for the second turbulent burst. The intensity of the Reynolds Stress is smaller in the three-dimensional duct simulation than in the three-dimensional channel simulation, as is the case with the overall turbulent intensity. There is, however, an increase in the Reynolds Stress in the centre of the duct. As noted in section 7.2, this increase in the Reynolds Stress in the centre of the duct is most likely a result of the averaging used to create figure 7.5.
Figure 7.5: Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 918.6$. The horizontal lines at $\pm0.80$ indicate the width of the Stokes boundary layer.
Figure 7.6: A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 7.0$, $Re_\delta = 884.77$. 

Chapter 7. Presentation of Three-Dimensional Duct Simulation Results
Chapter 7. Presentation of Three-Dimensional Duct Simulation Results

Figure 7.6 shows the energy spectra of the three-dimensional duct simulation over the development of the first turbulent burst. Again, the pattern is similar to that in the three-dimensional channel simulation. The main difference is that the amount of energy in the higher values of $k_d$ is significantly greater than that in the three-dimensional channel simulation. This suggests that the resolution of the lattice has a greater effect on the development of turbulence in the three-dimensional duct simulation than it does in the three-dimensional channel simulation. This is probably due to the periodic boundary condition in the $z$-direction creating an infinite channel. This always allows larger vortexes to be generated, which in turn changes the overall distribution of energy between the different length scales. This would also account for the larger turbulent intensity in the three-dimensional channel simulation.

7.3 Conclusions

The results for the three-dimensional duct simulations are very similar, as expected, to the three-dimensional channel simulations. There are differences in the dimensionality during development of the primary instability, but this is most likely caused by the increase of restrictions on the flow in a duct compared to that in a channel. As the primary instability develops, the individual vortexes formed are again two-dimensional in nature, but they are not all aligned in the same plane, as was the case in the three-dimensional channel simulations. This gives
the appearance that the vortexes formed in the primary instability of the three-dimensional duct simulations are three-dimensional in nature. However, combining the evidence from the three-dimensional channel and three-dimensional duct simulations supports the interpretation that the primary instability is two-dimensional in nature.
Chapter 8

Presentation of Particle Image Velocimetry Results

Here, the results of the three-dimensional duct simulation are compared to PIV measurements of turbulence development in oscillatory flow in a square pipe. The parameters of the flow from which the measurements were taken have been chosen so that the value of $Re_\delta$ and $\alpha$ are comparable with those in the simulation presented in Chapter 7. The PIV measurements were carried out in a square duct of width 8mm and length $\sim 1m$ before connections to the pump. When the experiment was driven by a 4V amplitude sine wave, the maximum flow velocity measured by PIV was $0.86ms^{-1}$. This equates to maximum value of the Reynolds number based on the Stokes boundary layer thickness of $657.14$, which is comparable to the maximum value of $Re_\delta$ in the simulation in chapter
7, which is 884.77. The maximum value of \( Re_5 \) in the experiments is significantly
smaller than than in the simulations and this will result in a reduction in the
turbulent intensity in the experimental results. However, both the experimental
and simulation results are well into the intermittently turbulent regime, table
2.1, and so should display the same development features. As it is impossible to
look at the development of the first turbulent burst in an experimental situation,
the focus here will be on the comparison of the second and subsequent bursts
as a validation of the lattice-Boltzmann simulations of the first turbulent burst.
There are several reasons that the first turbulent burst cannot be observed in
an experimental situation. The main one is that there is no way to initialise a
laminar flow with a turbulent Reynolds number, as is easily the case in simulation
by inserting a profile calculated from the analytical solution to the Navier-Stokes
equation. Also, it would be very difficult to repeat generation of the first turbulent
burst so precisely that, when the large number of averages required for PIV have
been taken, a coherent picture is still available.

8.1 Location of PIV Measurements

PIV is limited to taking a two-dimensional measurement of the velocity field
in a given flow. To build up an understanding of full three-dimensional picture of
the flow, multiple measurements need to be taken at different locations in the flow
for any given time. The PIV measurements made here have taken slices along the
Chapter 8. Presentation of Particle Image Velocimetry Results

Figure 8.1: Diagram showing example PIV slices taken through the experimental duct.

cross-stream direction, measuring the velocities in the $x - y$ plane. It is difficult to take measurements close to the boundaries of the cross-stream direction due to bonding of the walls of the perspex duct, so slices have been taken at 2, 3, 4, 5 and 6 mm from the near edge of the cross-stream direction. A sample layout of the slices is shown in figure 8.1. Measurements of the velocity field at each of these locations across the cross-stream direction were taken at eight phase points through the oscillatory cycle, at $\omega t = 0.785, 1.571, 2.356, 3.142, 3.927, 4.712, 5.498$ and 6.283.
8.2 Comparison of PIV measurement and Lattice-Boltzmann of a Turbulent Burst

Figure 8.2 shows the turbulent intensity over a full oscillatory cycle over the spanwise or y-direction of the duct. This can be compared directly to the turbulent intensity in the three-dimensional duct simulation in chapter 7. Figure 8.3 shows a map of the turbulent intensity over a whole oscillatory cycle showing the third and fourth turbulent bursts of the simulations shown in figure 7.2. To aid the direct comparison, data has only been taken from the simulation at the phases measured in the PIV experiments (section 8.1) In both figures 8.2 and 8.3, the phase numbering indicated on the $\omega t$-axis has a different meaning to that in similar plots in chapters 5, 6 and 7. In the preceding results chapters, the phase has indicated the time elapsed from the start of the simulation with a laminar flow initialised from the analytical solution to the Navier-Stokes equation for each flow situation. As the PIV measurements are 'snapshots' of a fully developed flow, the $\omega t$-axis only indicates the phase of the oscillation, and so only runs from 0 to $2\pi$ in figure 8.2. As $\omega t = 0$ and $\omega t = 2\pi$ are equivalent, figure 8.2 only plots data from $\omega t = \frac{\pi}{4}$ to $\omega t = 2\pi$. The $\omega t$-axis in figure 8.3 has been rescaled to reflect this. To return the scale to the context of the plots in chapter 7, $2\pi$ needs to be subtracted from the values on the $\omega t$-axis. The contour levels in figure 8.3 have also been rescaled so that they match those in figure 8.2 to aid in a direct comparison.
Figure 8.2: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $c = 6.76, \text{Re}_\delta = 824$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
Figure 8.3: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 7.0$, $Re_\delta = 824$. The horizontal lines at $\pm0.80$ indicate the width of the Stokes boundary layer.
The turbulent intensities shown in figures 8.2 and 8.3 show that the main features of turbulence development over one oscillation in a developed flow are broadly similar. In both the PIV measurements and the three-dimensional duct simulation, there are two separate turbulent bursts separated by a relaminarisation of the flow. In both cases, the turbulence is most intense in the Stokes boundary layer and appears to develop in this area before moving into the centre of the channel.

The most obvious difference between the simulation and PIV plots is that the turbulent intensity in the simulation, figure 8.3, is larger in magnitude, particularly in the second half of the oscillation. There are several reasons for this. The first is that the Reynolds number based on the Stokes boundary layer width, $Re_\delta$, for the simulation is higher than that in the PIV measurements, with $Re_\delta = 884.77$ for the simulations compared with $Re_\delta = 657.14$ for the PIV measurements. This means that the flow in the simulations will generate more turbulence. The larger value of $Re_\delta$ in the simulation will also mean that the turbulence takes longer to die away in the simulation. As the PIV measurements only measures the streamwise and spanwise components of the velocity, turbulent velocities in the cross-stream direction are ignored by the PIV measurements. Aside from the difference in the intensity of the turbulent bursts, there is a very similar pattern in the generation of turbulence in both the simulation and the PIV plots. Both sets of results show the largest turbulent intensities in the Stokes boundary layer, with the turbulent intensity generally increasing in this area of
Chapter 8. Presentation of Particle Image Velocimetry Results

the channel before developing in the centre of the duct. The major difference in the pattern of turbulence development if the phase at which the minimum turbulent intensity occurs. In the PIV measurements, figure 8.2, the minimum turbulent intensity at the end of the first turbulent burst occurs when \( \omega t = 3.142 \). In the simulation, it occurs when \( \omega t = 3.927 \), figure 8.3. The difference in the phase of the minimum intensity between the PIV measurements and the simulation could be accounted for by the lower intensity of the burst in the PIV measurements, as a lower intensity burst will die away more quickly, allowing the flow to re-laminarise to a greater extent and the development of the next burst to dominate in the boundary layer earlier in the phase. In the first burst shown in figure 8.2, there is also a smaller increase in the centre of the channel as the burst is developing, at \( \omega t = 0.942 \). This occurs for the same reason as the increase in the centre of the channel seen in figure 7.2 and discussed in section 7.2. As this is square duct, rather than a channel, there is also a Stokes boundary layer at the edges of the cross-stream direction and the turbulent intensity will increase here before it begins to increase in the main flow. The fact that this increase in the Stokes boundary layer is only visible in the centre of the channel indicates that the generation of turbulence is greater along the centre of the wall than it is in the corners of the duct. The second burst shown in figure 8.2 is significantly smaller in magnitude than the first, particularly in the Stokes boundary layer. There could be several reasons for this. The most likely is that seen in chapter 5 for the two-dimensional simulations. There, the magnitude of a given burst is influenced
by the burst which preceded it. So a burst with a large intensity is followed by a burst with a smaller intensity. Figure 8.4 shows a plot of the normalised mean velocities, compared to the normalised force for both the simulation and PIV measurements. As in figure 5.4 for the two-dimensional simulations, the driving force leads the mean velocity by a phase $\sim \frac{\pi}{2}$. The phase of the mean velocity for the PIV measurements and simulation will be slightly different, due to the difference in the value of $\alpha$, but this should not be a significant factor. Comparing the mean velocity between the simulations and PIV measurements, there is no significant difference within the limitations of the measurements. It is therefore sensible to assume that the peak velocity measured in the PIV experiments may not be the absolute peak velocity of the flow. This can be seen in figure 8.4, particularly at $\omega t = 1.571$, where the peak of the PIV velocity occurs significantly before the peak of the LBM velocity. While this indicates that the maximum velocity, and therefore the maximum Reynolds number based on the Stokes boundary layer, may be higher than has been measured in the PIV experiments, it is reasonable to assume that it is still smaller than that of the simulation. This can be seen in figure 8.3, where the turbulent intensity for the simulation is generally higher at all times.

A further difference between the PIV measurements and the three-dimensional simulations is the difference in the development of the flow. In the PIV measurements, the fluid has made a large number of oscillations before the measurements are taken. This is in contrast to the simulations, where the measurements are
Figure 8.4: Contour plot of the normalised turbulent intensity, $q^*$ across the width of the channel width $\omega t$ with $\alpha = 6.76$, $Re_\delta = 824$. The Stokes boundary layer is not visible, due to its small width in the experimental situation.
taken within one or two oscillations from a laminar flow. This is because it is not practically possible to run the simulations for more than two complete oscillations. It is impossible to make PIV measurements on a flow that is only two oscillation from a laminar flow. As each turbulent burst is affected by the preceding one, it is reasonable to assume that the bursts may vary significantly in the first few oscillations, as in the simulations, before becoming more settled after a large number of oscillations, as in the PIV measurements.

8.2.1 Reynolds Stress and Turbulent Energy Cascade

Further similarities between the PIV measurements and the two-dimensional simulations are evident in the plot of the normalised Reynolds stress, figure 8.5. The normalised Reynolds stress has a far greater magnitude than any of the simulations. For all three different lattice-Boltzmann simulations, $\varphi^* \sim 10^{-4}$. In the PIV measurements, $\varphi^* \sim 1$. However the development of the normalised Reynolds stress over the oscillatory cycle bears similarities with the two-dimensional simulations. Comparing figure 8.5 to the plot of the normalised Reynolds stress for the three-dimensional duct simulation, figure 5.10, shows that in the three-dimensional duct simulation, the normalised Reynolds stress appears mainly in the Stokes boundary layer, where as in the PIV measurements, it appears across the full width of the spanwise direction. Although the appearance of a significant Reynolds stress across the whole spanwise direction is similar to that observed
in the two-dimensional simulation, there are some important differences. The Reynolds stress appears in a much less structured fashion in the PIV measurements, with areas of high Reynolds stress magnitude followed by areas of low magnitude across the spanwise direction at the same phase in the oscillatory cycle. This is distinctly different from the situation in the two-dimensional simulations, figure 5.10, where the Reynolds stress occurs continuously across the spanwise direction. This is most clearly illustrated at $\omega t \sim 3.142$, where the Reynolds stress occurs from the edge of the Stoke boundary layer at $y \sim -0.8$ to the centre of the channel with a positive magnitude and then continues from the centre of the channel to the edge of the Stoke boundary layer at $y \sim 0.8$ with a negative magnitude. It has been stated that higher order correlations, such as Skewness and Kurtosis, are so sensitive to noise in PIV measurements that their usefulness is severely limited[20]. It is possible that this also accounts for the differences between the structure of the Reynolds Stress seen in these PIV measurements and that seen in the simulations and previous PIV measurements[73].

The turbulent energy spectra for the PIV measurements does not really reveal anything about the flow due to the lack of resolution in the PIV measurements. Due to the small value of the Kolmogorov length, with $l_k \sim 10^{-4}$, the resolution of data points in the PIV measurements does not provide enough frequency components to provide spectral components in the inertial range. To increase the resolution of the PIV measurements to provide enough velocity vectors to calculate the energy spectra into the range $0.1 < k_d < 1$. 

154
Figure 8.5: Contour plot of the normalised mean Reynolds stress, $\varphi^*$, across the width of the channel width $\omega t$ with $\alpha = 6.76$, $Re_\theta = 824$. The horizontal lines at $\pm 0.80$ indicate the width of the Stokes boundary layer.
Figure 8.6: A plot of the turbulent energy spectra, $E(k)$ at different values of $\omega t$, indicated in the legend, with $\alpha = 6.76$, $Re_\delta = 824$. 
8.3 Conclusions

The PIV measurements compare favourably with the three-dimensional duct simulation. Although the PIV measurements are lacking in detail, the phase and magnitude information obtained compares closely with the results from the three-dimensional duct simulation. The overall magnitude of the turbulence in the PIV measurements is smaller than in the three-dimensional duct simulation, but there are two factors influencing this. The first is that the Reynolds number based on the Stokes boundary layer for the PIV measurements is smaller than the simulation. It follows, therefore, that the turbulent intensities in the PIV measurements will be smaller also. The second reason is the phase in the oscillation where the PIV measurements were made, shown in figure 8.4. As explained in section 8.2, the peak velocity measured here is likely not to be the peak velocity of the flow. All the simulations point to the maximum turbulent intensity occurring just after the peak flow velocity. This indicates that the maximum turbulent intensity in the PIV measurements is higher than the values measured.
Chapter 9

Conclusions and Future Work

9.1 Conclusions

The main aim of this thesis has been to develop a three-dimensional lattice-Boltzmann model which can simulate bounded oscillatory flows into the turbulent regime. It has clearly been shown that the LBM is an effective tool in simulating this type of flow, both in the production of a bounded oscillatory flow in the turbulent regime and when the three-dimensional simulation was compared to the PIV measurements.

The simulations are not perfect however with the main problem being a lack of resolution at small scales. While this obviously reduces the accuracy of the results, the overall trends match closely those from the PIV measurements. The
additional information on the flow which can be obtained from computational simulations allows a greater understanding of the development of turbulence than that which can be obtained from experimental measurements as the mechanism by which the turbulence develops can be more closely observed. While this is true for all computational techniques, the LBM provides an accurate and efficient direct numerical simulation in real space over a significant proportion of the important length scales. This could be further improved by an increase in the lattice resolution.

Although the broad features of the three-dimensional simulation compare well with the PIV measurements taken, the accuracy of the PIV measurements needs to be increased to provide a fuller comparison of the complex features of the flow, particularly during the development of turbulence.

9.2 Future Work

The work carried out for this thesis could be extended in both the experimental and computational directions.

9.2.1 Extension of Computational Simulations

The simulations in this study could be extended in two distinct ways. Firstly, further simulations need to be carried out in the three-dimensional duct simu-
lution for different parameters, most importantly several different values of the Womersley parameter, $\alpha$. The second extension is an increase in the lattice resolution, which would reduce the problems of the low resolution at small scales. However, increasing the lattice resolution also means a huge increase in the runtime of the simulations. As well as the increase in the number of lattice sites as the lattice resolution increases, the number of timesteps required for an oscillation scales as a square of the total number of lattice sites in the duct diameter to maintain the same values of $\alpha$ and $Re_d$. To be able to run these larger simulations in a practical time frame would require some changes to the simulation code. The most important of these would be a change in the way the larger number of lattice sites would be split over the processors in a parallel machine for a simulation with a higher lattice resolution. This change is required to allow a larger number of processors to work most efficiently. Increases in computing technology, even since this work was started, also mean that it is already possible to investigate flows with a much wider range of parameters by using the same techniques on newer, faster hardware. If the number of computing nodes available were to be increased, it would be possible to increase the lattice resolution. This would require the reorganisation of the lattice decomposition to minimise the surface area to volume ratio of the lattice sites on each computing node, so as to minimise the communication time between the computing nodes at each timestep during the simulation. Presuming that a approximately linear speed up could be achieved, a doubling in the number of lattice dimensions in each direc-
tion coupled with an increase in the number of computing nodes to 256 would allow the three-dimensional duct simulation from Chapter 7 to be reproduced in a similar runtime.

9.2.2 Extension of PIV Measurements

The main problem with the PIV measurements in this study is a lack of resolution. This could be rectified with a combination of two steps. The first is to change the parameters of the flow to increase the width of the duct, while keeping the same values for \( \alpha \) and \( Re_s \). The second is to use a higher resolution digital camera to increase the amount of data captured in the images. If the width of the duct used is increased, this could mean that the amount of light provided by the copper vapour laser used in this study would not be sufficient, and so a more powerful laser would be required. An increase in the duct width would also require a more powerful pump to drive the flow to a high enough Reynolds number. Increasing the resolution of the camera would most likely require a change in the seeding particles used in the measurements, as their size relative to the integration area would need to be maintained. These factors mean that the best way to increase the resolution would be a combination of increasing both the width of the duct and the resolution of the camera. In terms of the energy spectra of the PIV measurements, the number of velocity vectors in the streamwise direction could be increased with the use of a longer duct and
multiple cameras. These improvements in the experimental setup would provide information on the higher modes of the energy spectra.
Chapter 10

Publications

10.1 Journal Papers


10.2 Book Chapters


10.3 Conference Publications


Bibliography


[27] Shiyi Chen Hudong Chen and William H. Matthaeus. Recovery of the navier-
   stokes equations using a lattice-gas boltzmann method. *Physical Review A*, 


170


