Numerical simulation of a flag behind a flat plate in a uniform flow

2019

A thesis submitted to The University of Manchester for the degree of Doctor of Philosophy in the Faculty of Science and Engineering.

By

JONATHAN A. SANCHEZ MUNOZ

SCHOOL OF MECHANICAL, AEROSPACE AND CIVIL ENGINEERING
1 Introduction

1.1 Motivation

1.2 Objectives of this work

1.3 Scope

1.4 Organization of the thesis

2 Literature Review

2.1 Flow control

2.1.1 Passive drag reduction techniques

2.2 Flapping and bending bodies

2.3 Arrays of flapping and bending bodies

2.4 Bluff body attached to a flexible filament

2.5 Summary of chapter 2

3 Lattice Boltzmann Method

3.1 Introduction

3.2 The Navier Stokes equations

3.2.1 Navier-Stokes solvers

3.3 Lattice Boltzmann solvers

3.4 Boltzmann's background and kinetic theory

3.5 Maxwell Boltzmann distribution
# TABLE OF CONTENTS

3.6 Lattice Boltzmann Method ................................................. 45
   3.6.1 Boltzmann Transport equation ..................................... 46
   3.6.2 Bhatnagar,Gross and Krook (BGK) Approximation ................. 47
   3.6.3 Lattice Boltzmann Model ........................................... 48
   3.6.4 Lattice Boltzmann method implementation ......................... 50
3.7 Hydrodynamic forces over solid objects and complex geometries ....... 50
   3.7.1 Boundary conditions for an immersed body in a uniform flow .... 51
   3.7.2 Staircase approximation ........................................... 51
   3.7.3 Non-conforming meshes ............................................ 54
3.8 Lattice Boltzmann Units ................................................... 56
   3.8.1 Units conversion and rules to follow ............................... 56
   3.8.2 Non-Dimensional units to Lattice Units ............................. 57
3.9 From the Lattice Boltzmann equations to the Navier-Stokes equations .. 59
3.10 Lattice Boltzmann solvers vs Navier-Stokes solvers ...................... 60
3.11 Summary chapter 3 ....................................................... 62

4 Message Passing Interface (MPI)-Lattice Boltzmann method and mesh refinement study 63
   4.1 Introduction ........................................................... 63
   4.2 Hardware performance study .......................................... 63
      4.2.1 Hardware selection ............................................... 63
   4.3 2D Flat plate experiments ........................................... 64
   4.4 3D Flat plate experiments ........................................... 66
   4.5 Summary chapter 4 ..................................................... 67

5 Validation test case: Bluff body ............................................. 69
   5.1 Cylinder immersed in a cross flow ................................... 69
      5.1.1 Pressure coefficient validation $C_p$ .......................... 70
      5.1.2 Drag coefficient and Strouhal number ......................... 74
      5.1.3 Bubble size behind cylinder ................................... 75
      5.1.4 Lift coefficient and Vorticity ................................ 76
   5.2 Flat plate immersed in a uniform flow ................................ 78
      5.2.1 Pressure Coefficient over a flat plate .......................... 78
      5.2.2 Drag and Lift coefficient over a flat plate .................... 81
   5.3 Summary of chapter 5 ................................................ 84

6 Validation test case: Flexible flag ....................................... 87
   6.1 Euler-Bernoulli beam theory .......................................... 87
   6.2 Flexible flag immersed in a cross flow ............................. 88
# TABLE OF CONTENTS

6.2.1 Instantaneous analysis ........................................ 89
6.2.2 Tail position analysis ........................................ 90
6.2.3 Bifurcation behaviour ........................................ 94
6.3 Summary of chapter 6 ........................................... 95

7 Proposed experiments and discussion .......................... 97
7.1 Bifurcation behaviour analysis, Reynolds Re=100,200,400 .... 97
  7.1.1 Reynolds Re=100, Mass ratio $U_s=20-100$ and Reduced velocity $\mu=0.05-10$ . 98
  7.1.2 Reynolds Re=200, Mass ratio $U_s=20-100$ and Reduced velocity $\mu=0.05-10$ . 99
  7.1.3 Reynolds Re=400, Mass ratio $U_s=20-100$ and Reduced velocity $\mu=0.05-10$ 100
7.2 Flexible flag behind a flat plate ................................ 102
  7.2.1 Flapping frequency analysis ................................ 103

7.3 Tail displacement analysis .................................... 106
  7.3.1 $Re=200$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$ ............... 107
  7.3.2 $Re=400$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$ ............... 110

7.4 Pressure coefficient $C_p$ ....................................... 113
  7.4.1 Pressure coefficient, $Re=200$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$ ............... 113
  7.4.2 Pressure coefficient, $Re=400$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$ ............... 115

7.5 Drag coefficient, ................................................. 115
  7.5.1 Drag coefficient, $Re=200-400$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$ ............... 117

7.6 Summary chapter 7 ............................................... 118

8 Conclusions and future work .................................... 121
  8.1 Future Work ..................................................... 124

A Appendix A ............................................................ 125
  A.1 Introduction ....................................................... 125
  A.2 A brief history of MPI .......................................... 125
  A.2.1 MPI program structure ....................................... 126
  A.2.2 Communication modes: Point to point or collective communications .... 126
  A.3 Lattice Boltzmann Method and MPI .......................... 128
  A.3.1 Lattice Boltzmann Method and MPI decomposition ........... 129
  A.4 Refinement Region method ..................................... 130
  A.4.1 Local embedded uniform grid ................................. 131
  A.5 Summary appendix A ............................................. 134

Bibliography ........................................................... 135
<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1 Splitter plates experiments</td>
<td>26</td>
</tr>
<tr>
<td>2.2 Flapping and bending bodies experiments</td>
<td>27</td>
</tr>
<tr>
<td>2.3 Array of flexible or bending bodies</td>
<td>32</td>
</tr>
<tr>
<td>2.4 Array of flexible or bending bodies</td>
<td>33</td>
</tr>
<tr>
<td>3.1 Comparative between open source and commercial CFD software</td>
<td>40</td>
</tr>
<tr>
<td>3.2 CFD solvers available with Lattice Boltzmann methodology simulations</td>
<td>42</td>
</tr>
<tr>
<td>4.1 Characteristic length of lattice units for each level</td>
<td>64</td>
</tr>
<tr>
<td>4.2 Hardware test experiments 2D flat plate</td>
<td>65</td>
</tr>
<tr>
<td>4.3 Characteristic length of lattice units for levels</td>
<td>66</td>
</tr>
<tr>
<td>4.4 Hardware test experiments on a 3D flat plate</td>
<td>67</td>
</tr>
<tr>
<td>5.1 Set up in lattice units for circular cylinder experiment</td>
<td>71</td>
</tr>
<tr>
<td>5.2 Set up of experiments in lattice units for flat plate</td>
<td>79</td>
</tr>
<tr>
<td>A.1 MPI history</td>
<td>126</td>
</tr>
<tr>
<td>A.2 MPI Implementations</td>
<td>126</td>
</tr>
<tr>
<td>A.3 MPI Communicators</td>
<td>127</td>
</tr>
<tr>
<td>A.4 MPI Communicators</td>
<td>128</td>
</tr>
<tr>
<td>A.5 MPI Communicators</td>
<td>128</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>2.1</td>
<td>24</td>
</tr>
<tr>
<td>2.2</td>
<td>25</td>
</tr>
<tr>
<td>2.3</td>
<td>26</td>
</tr>
<tr>
<td>2.4</td>
<td>30</td>
</tr>
<tr>
<td>3.1</td>
<td>39</td>
</tr>
<tr>
<td>3.2</td>
<td>41</td>
</tr>
<tr>
<td>3.3</td>
<td>41</td>
</tr>
<tr>
<td>3.4</td>
<td>43</td>
</tr>
<tr>
<td>3.5</td>
<td>45</td>
</tr>
<tr>
<td>3.6</td>
<td>46</td>
</tr>
<tr>
<td>3.7</td>
<td>49</td>
</tr>
<tr>
<td>3.8</td>
<td>49</td>
</tr>
<tr>
<td>3.9</td>
<td>50</td>
</tr>
<tr>
<td>3.10</td>
<td>51</td>
</tr>
<tr>
<td>3.11</td>
<td>52</td>
</tr>
<tr>
<td>3.12</td>
<td>52</td>
</tr>
<tr>
<td>3.13</td>
<td>53</td>
</tr>
<tr>
<td>3.14</td>
<td>55</td>
</tr>
<tr>
<td>3.15</td>
<td>60</td>
</tr>
<tr>
<td>4.1</td>
<td>65</td>
</tr>
<tr>
<td>4.2</td>
<td>66</td>
</tr>
<tr>
<td>4.3</td>
<td>67</td>
</tr>
<tr>
<td>5.1</td>
<td>70</td>
</tr>
<tr>
<td>5.2</td>
<td>71</td>
</tr>
<tr>
<td>5.3</td>
<td>72</td>
</tr>
<tr>
<td>5.4</td>
<td>73</td>
</tr>
<tr>
<td>5.5</td>
<td>73</td>
</tr>
<tr>
<td>5.6</td>
<td>74</td>
</tr>
<tr>
<td>5.7</td>
<td>75</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

5.8 Bubble scheme. .................................................. 75
5.9 Length of time averaged separation bubble behind cylinder vs Reynolds number. ....... 76
5.10 Time average velocity , Re=4,20,60 and 160 .................................................. 76
5.11 Instantaneous lift coefficient over time. .................................................. 77
5.12 Vorticity , Re=4,20,60 and 160 .................................................. 78
5.13 Configuration of flat plate immersed in a cross flow .................................. 79
5.14 Pressure coefficient over flat plate, $\alpha=0-20$ degrees. .................................. 80
5.15 Time average pressure, Re=300 , $\alpha=20,30,40$ and 60 ............................... 80
5.16 Time Average velocity , Re=300 , $\alpha=20,30,40$ and 60 .................................. 81
5.17 Drag and Lift coefficient for $\alpha=0-60$ degrees. ........................................ 81
5.18 Cd and Cl over time of Taira and Colonious for a five different angles and Re=300. .... 82
5.19 Cd vs Time, $\alpha=0-60$ .......................... 83
5.20 Cl vs Time, $\alpha=0-60$. .................................................. 83
5.21 Vorticity, Re=300 , $\alpha=20,30,40$ and 60 .................................................. 84

6.1 Flexible filament immersed in a cross flow. ............................................ 89
6.2 Proposed experiments for bifurcation behaviour analysis. ................................ 89
6.3 Configuration of the experiment, flexible flag. ........................................... 90
6.4 Tail position analysis ( from top to bottom ,$\mu=0.1,0.2,0.8,2.0,3.0$ ) ,a) Y displacement, b) Tail velocity $dy/dt$ and c) Normalized power spectra. .................................. 91
6.5 Time-evolving vorticity contours of the flow surrounding the flag for $\mu=0.2,Re=200$ and $K_b=0.0001$. .................................................. 92
6.6 Time-evolving vorticity contours of the flow surrounding the flag for $\mu=0.8,Re=200$ and $K_b=0.0001$. .................................................. 93
6.7 Time-evolving vorticity contours of the flow surrounding the flag for $\mu=2.0,Re=200$ and $K_b=0.0001$. .................................................. 93
6.8 Time-evolving vorticity contours of the flow surrounding the flag for $\mu=3.0,Re=200$ and $K_b=0.0001$. .................................................. 94
6.9 Bifurcation behaviour Re=200. .................................................. 95

7.1 Proposed experiments to identify limits among different flapping states. .......... 98
7.2 Bifurcation behaviour Re=100. .................................................. 99
7.3 (Left) Flapping frequency,(Right)Flapping amplitude for selected mass ratios $\mu$ and reduced velocities $U_s$ .................................................. 99
7.4 Bifurcation behaviour Re=200. .................................................. 100
7.5 (Left)Flapping frequency, (Right)Amplitude for selected mass ratios $\mu$ and reduced velocities $U_s$ .................................................. 100
7.6 Bifurcation behaviour Re=400. .................................................. 101
7.7 (Left) Flapping frequency, (Right) Amplitude for selected mass ratios $\mu$ and reduced velocities $U_s$ .......................................................... 101
7.8 Flat plate and flexible flag configuration. .......................................................... 102
7.9 Set of experiment proposed. ............................................................................ 103
7.10 Flapping frequency $\alpha=20$, Re=100-400, $U_s=5-30$ and $\mu=0.05-0.3$. ........ 104
7.11 Flapping frequency $\alpha=30$, Re=100-400, $U_s=5-30$ and $\mu=0.05-0.3$. ........ 105
7.12 Flapping frequency $\alpha=40$, Re=100-400, $U_s=5-30$ and $\mu=0.05-0.3$. ........ 105
7.13 Instantaneous vorticity flat plate $\alpha=40$, Re=400 ............................................. 106
7.14 Instantaneous vorticity flat plate and rigid flexible flag $\alpha=40$ ......................... 107
7.15 Tail displacement analysis, Re=200, $\alpha=40$, $U_s=5-30$ (top-bottom) and $\mu=0.1$ 108
7.16 Instantaneous vorticity Re=200, $\alpha=40$, $U_s=5$ and $\mu=0.1$ ......................... 108
7.17 Instantaneous vorticity Re=200, $\alpha=40$, $U_s=10$ and $\mu=0.1$ ....................... 109
7.18 Instantaneous vorticity Re=200, $\alpha=40$, $U_s=20$ and $\mu=0.1$ ....................... 109
7.19 Instantaneous vorticity Re=200, $\alpha=40$, $U_s=30$ and $\mu=0.1$ ....................... 110
7.20 Tail displacement analysis, Re=400, $\alpha=40$, $U_s=5-30$ (top-bottom) and $\mu=0.1$ 111
7.21 Instantaneous vorticity Re=400, $\alpha=40$, $U_s=5$ and $\mu=0.1$ ......................... 111
7.22 Instantaneous vorticity Re=400, $\alpha=40$, $U_s=10$ and $\mu=0.1$ ....................... 112
7.23 Instantaneous vorticity Re=400, $\alpha=40$, $U_s=20$ and $\mu=0.1$ ....................... 112
7.24 Instantaneous vorticity Re=400, $\alpha=40$, $U_s=30$ and $\mu=0.1$ ....................... 113
7.25 Time average pressure, Re=200, $\alpha=40$ ..................................................... 114
7.26 Pressure coefficient Re=200, $\alpha=40$, $U_s=5-30$, $\mu=0.1$ ............................. 115
7.27 Time average pressure, Re=400, $\alpha=40$ ..................................................... 116
7.28 Pressure coefficient Re=400, $\alpha=40$, $U_s=5-30$, $\mu=0.1$ ............................. 116
7.29 Instantaneous drag coefficient Re=200 and $\alpha=40$, $U_s=5-30$, $\mu=0.1$ .......... 117
7.30 Instantaneous drag coefficient Re=400 and $\alpha=40$, $U_s=5-30$, $\mu=0.1$ .......... 117
7.31 Averraged drag coefficient vs Reduced velocity. ............................................ 118

A.1 MPI program structure. ..................................................................................... 127
A.2 Split domain into blocks. .................................................................................. 129
A.3 MPI assignation. ............................................................................................... 129
A.4 Identify neighbours. ......................................................................................... 130
A.5 Swap information among processes. ................................................................. 130
A.6 Embedded grids. ............................................................................................... 131
A.7 Collision step coarse and fine region. ............................................................... 131
A.8 Homogeneous redistribution from coarse to finest regions. ............................. 132
A.9 Streaming step coarse and fine grids. ............................................................... 132
A.10 Collision step on fine grid. ............................................................................... 133
A.11 Stream coarse and fine grids. .......................................................................... 133
A.12 Homogeneous redistribution of particle density from fine to coarse grids. ....... 134
Abstract

The dynamic response of a flexible flag in a uniform cross flow was studied using the Lattice Boltzmann Method combined with the Immersed Boundary Method. In the first instance the flag was alone, while subsequently it was attached to the aft of a rigid flat plate, positioned at various angles of incidence to the flow. Two different boundary conditions were used to describe the objects; the rigid flat plate was defined via the staircase approximation while the flexible flag was simulated using the immersed boundary method. Prior to the examination of the aforementioned cases a significant amount of code testing was undertaken towards the release of an in-house code, LUMA (Lattice Boltzmann The University of Manchester) which has now been released via an academic journal publication in SoftwareX (2018). A mesh refinement study and a hardware performance analysis were completed in order to identify the most optimal use mode for this study. The flexible flag performance can be controlled through three main parameters Reynolds number, Re, reduced velocity $U_s$ and mass ratio $\mu$. The incidence angle $\alpha$ of the flat plate is introduced as the new control parameter. The combination of these parameters modify the dynamical states of the flexible flag as well as downstream flow. By varying these parameters over the ranges ($100 \leq Re \leq 400$), ($20 \leq \alpha \leq 40$), ($5 \leq U_s \leq 30$) and ($0.05 \leq \mu \leq 0.1$); we have identified four dynamical states, of which only three were previously observed in the literature. I) A regular flapping state wherein the flexible flag displaces uniformly with very small amplitude, and does not modify its shape. II) A pseudo-steady state where the flexible flag remains trapped by vortex shedding of the flat plate. III) A regular high displacement flapping state, whereby the flexible flag displaces uniformly. IV) An irregular flapping state, where the flexible flag describes a highly unsteady displacement cycle of an aperiodic nature. The dynamical states were compared with a rigid flag to quantify drag reduction and comparisons were made to configurations with rigid body counterparts. In cases where the dynamic nature of the flexible filament appeared to lock-in to the frequency of the unsteady shedding was reduced, in some cases resulting in drag reduction.
Declaration

No portion of the work referred to in the thesis has been submitted in support of an application for another degree or qualification of this or any other university or other institute of learning.
Copyright

i. The author of this thesis (including any appendices and/or schedules to this thesis) owns certain copyright or related rights in it (the "Copyright") and s/he has given The University of Manchester certain rights to use such Copyright, including for administrative purposes.

ii. Copies of this thesis, either in full or in extracts and whether in hard or electronic copy, may be made only in accordance with the Copyright, Designs and Patents Act 1988 (as amended) and regulations issued under it or, where appropriate, in accordance with licensing agreements which the University has from time to time. This page must form part of any such copies made.

iii. The ownership of certain Copyright, patents, designs, trademarks and other intellectual property (the "Intellectual Property") and any reproductions of copyright works in the thesis, for example graphs and tables ("Reproductions"), which may be described in this thesis, may not be owned by the author and may be owned by third parties. Such Intellectual Property and Reproductions cannot and must not be made available for use without the prior written permission of the owner(s) of the relevant Intellectual Property and/or Reproductions.

iv. Further information on the conditions under which disclosure, publication and commercialisation of this thesis, the Copyright and any Intellectual Property and/or Reproductions described in it may take place is available in the University IP Policy (see http://documents.manchester.ac.uk/DocuInfo.aspx?DocID=24420), in any relevant Thesis restriction declarations deposited in the University Library, The University Librarys regulations (see http://www.library.manchester.ac.uk/about/regulations) and in The Universitys policy on Presentation of Theses.
Acknowledgements

I am grateful to my father and mother, Arturo and Gabriela, who have provided me their unconditional support in my life. I am also grateful to my brother and sister who have supported me along the way.

Thank you Lucero for your love, support and advice.

I would like to express my sincere gratitude to my advisor Dr. Alistair Revell and for the continuous support of my Ph.D study and related research, for his patience, motivation, and immense knowledge.

I would like to express my gratitude to Dr. Adrian Harwood for his guidance and feedback to use LUMA.

A very special gratitude goes out to Consejo Nacional de Ciencia y Tecnologia (CONACYT) for providing the funding for this work.
In this chapter, we present the motivation and objectives of this work. We will also give the structure of this document.

1.1 Motivation

Flow over an immersed body is a highly relevant topic in fluid mechanics due to its prevalence across many natural phenomena. Throughout time scientists and engineers have looked to nature for inspiration in attempts to mimic desirable characteristics that might reduce drag enhance stability, improve efficiency. For example, the micro-riblets presented in shark skin have shown considerable improvements in viscous drag reduction; bumps on the leading edge of whale flippers, which delay stall and improve hydrodynamics manoeuvrability performance or bird feathers, which have gained interest due to the role in their aerodynamics flow control.

In recent years there has been a drastic rise in the development and application of unmanned-airborne-vehicles (UAVs), which due to their scale operate at low Reynolds numbers \(10^3 - 10^5\) and they are thus subject to entirely different aerodynamic effects compared to commercial airliners. Smaller still, micro-UAVs, operate at even lower Reynolds number \(10^2 - 10^3\), sharing similar scale effects with small animals such as birds, fishes or insects. Characteristic of these creatures, the interaction of fluid dynamics and structural dynamics play a profound role. In recent decades, the fascinating interplay between vortex shedding and structural deformation has caught the attention engineers more than ever before, and the topic of ‘Fluid-Structure Interaction’ (FSI) is now central to many engineering disciplines.

The study of flexible flags immersed in a cross-flow is considered one of the most interesting phenomena to study due to its viability to be applied for many engineering applications. During last decade scientist have been investigated this phenomenon with the intention to understand the interaction between an immersed flag and surrounding flow. The study of flexible flags
is not new; however it was until 2000 when Zhang et al. [96] published his work where was studied experimentally interaction between flexible filaments or flag that held upstream and free downstream surrounded by a fluid flow. Zhang found two different dynamical states, first one, the filament is immobile and aligned to flow direction, in a second case, the filament executes a sinuous motion similar to the flapping of a flag by the wind. In the same way, Zhang showed that the length of the filament plays a vital role to observe the changes in the state of the filament. For shorter length conditions, the filament remains in the stretched-straight state while for larger length conditions, the flexible filament presents a regular displacement.

1.2 Objectives of this work

In this section, we describe the main objectives of our research.

The general objective of this project is analysed numerically the behaviour of the flexible flag attached to a flat plate and its influence on near wall zone and downstream flow.

The particular objectives are:

- Validate a completely new code.
- Demonstrate that it is possible to couple two bodies, a rigid flat plate, defined by the staircase approach and a flexible body, defined with the immersed boundary method.
- Identify possible states of the flexible flag under the action of downstream flow created by the flat plate.
- Simulate numerically the performance of a flexible flag attached to a flat plate.
- Validate model implemented in LUMA to calculate drag and lift (Momentum exchange).
- Validate Message Passing Interface decomposition model implemented in LUMA.
- Develop a Message Passing Interface study to identify the fastest configuration.
- Validate Embedded refinement method implemented in LUMA.
- Validate Fluid-Structure interaction implemented in LUMA (Immersed boundary method).
- Develop codes to analyse LUMA output data.

1.3 Scope

Due to the lack of studies where a flexible flag is attached to a bluff body different than a circular cylinder, this study pretends to understand at low Reynolds, the relation between a flat plate and flexible flag and the effect under surrounding flow.
1.4 Organization of the thesis

This thesis organisation is:

- Chapter 2 provides a review of the literature for flow control, including a focus on both passive and active methods. This chapter continues with the description of flapping and bending bodies; we will review models, configurations, parameters and reference documents.

- Chapter 3 provides a summary of Lattice Boltzmann Methodology. We discuss theory and numerical methods used to solve the Lattice Boltzmann Method, and in particular, we review the boundary conditions used in this methodology.

- Chapter 4 shows a set of experiments to identify the best Message Passing Interface (MPI) decomposition for our tests.

- Chapter 5 shows the validation of the numerical solver for two different cases with rigid walls; a circular cylinder and a flat plate at incidence to the oncoming flow. In both cases, we measure performance and compare with detailed reference results.

- Chapter 6 shows a validation for a fluid-structure interaction model implemented in the solver. In this chapter, we present a tail displacement frequency analysis and a bifurcation behaviour study to identify limits among different flapping states for three different Reynolds numbers.

- Chapter 7 shows a set of results where flexible bodies work as a passive flow control device; we measure parameters such as pressure coefficient and drag coefficient. We also review the dynamic response, and we explain downstream flow under parameters selected.

- Chapter 8 provides the conclusion and a list of suggested activities for future work.
In this chapter, we review the most relevant works related to flapping bodies and focus on mechanisms of flow control. Firstly, we describe active and passive methods for flow control. Secondly, we identify documents for a single flapping body, arrays of flapping bodies as well as some papers which report the interaction between bluff bodies and flapping bodies.

2.1 Flow control

Drag reduction has earned importance in recent years due to the transportation industry. Researchers and engineers have been looking for alternatives to minimise the effect of drag and energy consumption. Drag reduction techniques can divide into two main categories, active and passive control.

Active flow controls, which use micro and nanoscale sensors and actuators, obtain feedback and control from the system more efficiently, their implementation, however, is expensive. On the other hand, passive techniques do not require energy expenditure or any input from the user; they are cheaper in comparison with active methods.

Figure 2.1 presents a summary of different flow control mechanisms. These different techniques can include blowing, suction, base cavities, vortex generator, after body modifications, splitter plates, boat tailing, injecting additives or droplets of fibres or large eddy break up devices. The main goal of active and passive methods is to modify the flow around the body, to prevent separation and reduce wake and mass transfer into the boundary layer.

2.1.1 Passive drag reduction techniques

In this work, we are going to focus on the passive flow control mechanism. The passive flow control mechanism can be divided into control surfaces, bluff modifications and retrofits.
In the case of control surfaces, the main idea is to modify the surface to control the boundary layer. Amongst these techniques, we can find polymer drag reduction additives, the micro-morphology, the super-hydrophobic surface, the micro air bubbles, the heating wall, the vibrant, flexible wall and the composite drag reduction methods. Luo [63] presents a summary of the advantages and percentage of drag reduction of all these technologies. In his paper, Luo showed that such techniques as vibrant, flexible wall or composite are more effective as they reduce drag production by 50% while micro-morphology reduces it by 8-12%.

Bluff body modifications is another passive method. The primary goal of this technique is to modify the shape of the body to have a similar performance as streamlined bodies, which in turn adjusts downstream flow. Choi [25] shows a summary of the most effective passive methods; for example, helical strake, which reduces force fluctuations and drag reduction significantly. Other examples include segmented trailing edge and wavy trailing which produces a dislocation in the wake and increases base pressure. Choi comments in his review that stagnation faces, with rectangular, sinusoidal and hemispherical bumps, do not reduce drag due to the separation which occurs at the front edge. Another technique, summarised by Choi, is the use of the upper and lower trailing edge of the bluff body. This method increases the base pressure by 33%; however, this phenomenon strictly depends on the selected configuration.

Retrofits is the last alternative shown in figure 2.1. The main purpose of this technique is to suppress vortex shedding to reduce drag. Other techniques include flaps, splitter plates and boat tails.

In the case of flaps, there are several attempts and configurations, mainly with Ahmed bodies. Such the case of Beaudoin [19], who reported a reduction of 7% in control flow through a set of moving flaps that are fixed on every edge around the two rear flat surfaces of the model. Fourrie [41] also worked on an Ahmed body, but in his experiments he placed a deflector on the upper edge of the model reducing drag by 9%, depending on the deflector angle.
2.2. FLAPPING AND BENDING BODIES

On the other hand, splitter plates have emerged as an alternative to reduce drag and suppress vortex shedding. One of the most common configurations is a splitter plate located downstream of the flow bluff body. There are many different configurations, not just in body shape but also in the length of the splitter plate $L$, the distance from the splitter plate to a bluff body, as well as with the Reynolds numbers.

Table 2.1 presents a summary of some of the tests with splitter plates as a mechanism of wake control to reduce drag. Most of these works have reported a significant amount of drag reduction, mainly due to the proper selection of the body shape, Reynolds number $Re$, splitter plate length $L$ and $G$ splitter plate-bluff body distance. However, a central characteristic that differs from most of the works is the implementation of a rotating splitter plate. Assi [16] and Gu [44] presented a rotating plate able to move freely to investigate pressure, drag and lift forces. Both works reported a reduction from 30% to 90% depending on the free rotating angle $\delta$ and the splitter plate length $L$. Figure 2.2 shows a summary of the rotating splitter plate.

![Figure 2.2: Rotating splitter plate configuration.](image)

Even rotating splitter plates have presented drag reduction, researchers and engineers have also focused their attention on flapping and bending bodies acting as splitter plates to modify downstream flow. This mechanism is observed in nature, for example, feather birds or fish swimming.

Before attempting to characterise the performance of flapping or bending bodies used as a mechanism for drag reduction, it is necessary to understand the behaviour under a stream flow. For that reason, in the next section, we are going to discuss some of the most relevant papers related to flapping bodies.

2.2 Flapping and bending bodies

The study of flapping bodies immersed in a cross-flow is considered one of the most interesting phenomena to study due to its viability to be applied for many engineering applications. During the last decade, scientists have studied this phenomenon with the intention of understanding the interaction between the immersed filament and the surrounding flow. In literature, there are several possible names to define a flapping body in an axial flow; for example, flags, cables, flutter bodies, flexible filaments, hanging chain, slender body. Usually, the configuration used considers a flapping body which reacts to the force exerted by an axial flow, figure 2.3.
CHAPTER 2. LITERATURE REVIEW

<table>
<thead>
<tr>
<th>Author</th>
<th>Method</th>
<th>Geometry</th>
<th>Reynolds</th>
<th>Measurements</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gallieron [42]</td>
<td>Experimental</td>
<td>Ahmed body</td>
<td>1.0-1.6 x 10^6</td>
<td>Drag, side force and yawing moment</td>
<td>7-28% Drag reduction ~ positioning</td>
</tr>
<tr>
<td>You [94]</td>
<td>Simulation</td>
<td>Circular</td>
<td>100-160</td>
<td>Lift, drag, Strouhal</td>
<td>Noise sources modifications</td>
</tr>
<tr>
<td>Akilli [10]</td>
<td>Exp-PIV</td>
<td>Circular</td>
<td>6300</td>
<td>Time average flow data</td>
<td>Wake region change up L/D=1.0</td>
</tr>
<tr>
<td>Rathakrishnan [78]</td>
<td>Experimental</td>
<td>Rectangular</td>
<td>0.58-0.98 x 10^5</td>
<td>Drag</td>
<td>Angle zero % center SP more Drag reduction</td>
</tr>
<tr>
<td>Anderson [13]</td>
<td>Exp-hot wire</td>
<td>Circular</td>
<td>2700-46000</td>
<td>Shedding frequency ~ L SP</td>
<td>Four regions identified ~ 1D-8t</td>
</tr>
<tr>
<td>Mansingh [65]</td>
<td>Exp. Wind tunnel</td>
<td>Rectangular</td>
<td>0.35-1.15 x 10^3</td>
<td>Shedding frequency, base pressure &amp; drag</td>
<td>50% Drag reduction ~ gap length ~ L</td>
</tr>
<tr>
<td>Doolan [30]</td>
<td>Simulation</td>
<td>square</td>
<td>150</td>
<td>Strouhal &amp; Cd</td>
<td>Stk force over plate Reduced</td>
</tr>
<tr>
<td>Assi [16]</td>
<td>Experimental</td>
<td>Circular</td>
<td>up to 30000</td>
<td>Drag</td>
<td>Drag reduction about 60 %</td>
</tr>
<tr>
<td>Mat Ali [66]</td>
<td>Simulation</td>
<td>Square</td>
<td>150 &amp; 0.5D ≤L≤ 6D</td>
<td>Strouhal</td>
<td>Suitable L for Strouhal</td>
</tr>
<tr>
<td>Gu [44]</td>
<td>Exp-Wind tunnel</td>
<td>Circular</td>
<td>3.6 x 10^4</td>
<td>St &amp; Drag</td>
<td>30-90% ~ displacement angle δ and L</td>
</tr>
<tr>
<td>Yeon Hwang [53]</td>
<td>Simulation</td>
<td>Circular</td>
<td>100-160</td>
<td>Strouhal, Drag &amp; Lift</td>
<td>3.0-6 % ~ G/d</td>
</tr>
</tbody>
</table>

Table 2.1: Splitter plates experiments

![Figure 2.3: Configuration of a flapping body immersed in an axial flow.](image)

There have been several attempts to reproduce and understand the behaviour of a flapping body. One of the first experiments was presented by Paidoussis [73] in 1975 where he performed a theoretical analysis of the behaviour of a flexible cylinder in an axial flow. Paidoussis found that parametric instabilities can be observed at specific ranges of frequencies and amplitudes; these instabilities are associated with particular flow velocities. This work inspired the work developed by Coene [26]; he was one of the first to discuss the equation of motion for a flexible, slender structure and compare it with a set of experiments in a wind tunnel with paper strips.

Similarly, Triantafyllou has primarily studied the flapping body phenomena, (cables, fish locomotion, chains, slender structures). In his studies Triantafyllou has talked about hydrodynamic response to axial flow, stability analysis, drag forces and drag reduction [18], thrust generation [86], locomotion through vortex shedding [56], vortex-induced vibrations and vortex control [47].

However, it wasn’t until 2000, that Zhang [24] performed a set of experiments using a vertical filament immersed in soap. Zhang identified and classified two different states, stretched-state,
## 2.2. FLAPPING AND BENDING BODIES

<table>
<thead>
<tr>
<th>Author</th>
<th>Parameters</th>
<th>Configuration</th>
<th>Measurements</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Paidoussis (1975) (T)[73]</td>
<td>Tension,E Young modulus, φ,M mass</td>
<td>Flexible cylinder, Uniform flow</td>
<td>Frequencies &amp; Instability regions</td>
<td>instabilities – perturbation frequencies &amp; amplitudes</td>
</tr>
<tr>
<td>Coene (T) (1992) [26]</td>
<td>$\rho_s$, Cross section area S, Axial tension T, Flexural rigidity $EI$</td>
<td>2D slender body, Immersed in axial flow</td>
<td>Flutter paper Uniform flow</td>
<td>Flutter speed, frequency</td>
</tr>
<tr>
<td>Watanabe (2002) (E) [91]</td>
<td>$\mu=0.058-0.093, E=10^{-6}$, $10^{-4}$</td>
<td>Flutter paper Uniform flow</td>
<td>Hysteresis – fluid speed, mass ratio, rigidity</td>
<td></td>
</tr>
<tr>
<td>Zhu (S)(2002) [64]</td>
<td>$\mu=0.058-0.093$, $EI=10^{-6}$-$10^{-4}$</td>
<td>Flapping amplitude</td>
<td>Stability analysis</td>
<td></td>
</tr>
<tr>
<td>Farnell (2005)(S)[37]</td>
<td>$K_s=0.013$, $Re=250$</td>
<td>Flexible body, Uniform stream</td>
<td>$x$ &amp; $y$ displacement, frequency</td>
<td>Euler &amp; Lagrangian equation coupled</td>
</tr>
<tr>
<td>Schouveiler (E) (2005) [81]</td>
<td>$U=0-13.6$ m/s, Young's modulus E=2.5 MPa</td>
<td>Uniform flow, horizontal plate</td>
<td>Frequency, Cd, $U_c$</td>
<td>Different flapping modes</td>
</tr>
<tr>
<td>Shelley (2005)(E) [82]</td>
<td>Mass ratio $S=m_L/\rho dL$, fluid kinetic energy $U=0.5\rho dL^2/2$</td>
<td>Flexible flag Uniform flow (Water)</td>
<td>$y$ displacement &amp; frequency</td>
<td>Linear stability analysis</td>
</tr>
<tr>
<td>Argentina (2005) (T)[14]</td>
<td>$L$, Young modulus, $\nu$</td>
<td>2D flag, Immersed in axial flow</td>
<td>Frequency flapping</td>
<td>Linear stability analysis</td>
</tr>
<tr>
<td>Connell (2007) (S)[27]</td>
<td>$\mu=0.025-0.3$, $Re=0-5000$, bending rigidity $K_B=0.0001$</td>
<td>2D flag, Uniform stream</td>
<td>Displacement, frequency, tail trajectory</td>
<td>Linear stability analysis</td>
</tr>
<tr>
<td>Huang(S) (2007)[49]</td>
<td>$K_B=0.0001$, $Re=200$, bending coefficient $\nu=10^{-4}$-$10^{-2}$</td>
<td>Flexible filament, Uniform flow</td>
<td>Flapping amplitude</td>
<td>Eulerian-Lagrangian grid, stability analysis</td>
</tr>
<tr>
<td>Paraz (E) [80]</td>
<td>Bending rigidity $L$, velocity $v$</td>
<td>Horizontal flexible plate, Uniform flow</td>
<td>Vortex shedding visualizations</td>
<td>Reversed sense of rotation of the vortices</td>
</tr>
<tr>
<td>Michelin(S)(2008)[68]</td>
<td>$L$, $B$, $M$</td>
<td>2D flexible flag, Axial flow</td>
<td>Power spectra, tail position, frequency, flapping mode</td>
<td>Hysteresis behavior</td>
</tr>
<tr>
<td>Favier (S) (2014) [39]</td>
<td>$Re=200$, $b_L=0.001$, $\mu=1.5$</td>
<td>2D flexible filament, Uniform flow</td>
<td>$y$ displacement, flapping pattern</td>
<td>couple fluid / filament dynamics</td>
</tr>
</tbody>
</table>

Table 2.2: Flapping and bending bodies experiments.

where the filament keeps aligned in the flow direction of the flow and the second state, known as the flapping state, where it produces a sinuous motion similar to a flapping flag.

Over the last 20 years, there have been many experimental, theoretical and numerical simulations which have tried to explain fluid flow and body interaction. Table 2.2 shows a summary of relevant works; here we will identify the coincidence in relevant parameters, configuration, methodology as well as the most important findings.

Inspired by Zhang’s work, Watanabe presented a set of papers [89], [90], [91] to analyse the flutter of a paper sheet. He published a numerical simulation and potential flow analysis of paper flutter through the Navier-Stokes equations. In his works, Watanabe tried to clarify the phenomenon of a fluttered body. He analysed the unsteady lift force, the amplitude of flutter as well as the performance of the flow around the flutter body. Watanabe compared various methods to analyse sheet flutter as presented by Noguchi [71], Huang [48], Yamaguchi [93], Kornecki [58] and Guo and Paidoussis [45]. He selected the potential flow analysis to analyse domain frequency,
and he was able to perform a parametric study to relate flutter mode with the mass ratio.

It wasn’t until 2003 that Zhu and Peskin [97] reported the numerical simulation of a flexible flapping filament in a flowing soap film. Based on Peskin’s [76] research about immersed boundary methods, they describe the behaviour of the filament and the fluid through the incompressible viscous Navier-Stokes equations. The formulation used by Zhu and Peskin introduced a Eulerian grid to describe fluid behaviour, while a Lagrangian grid was used to compute and describe flexible filaments. They reported that flapping of the filament is just possible if the mass is included in the model. The length of the filament was found to be an essential parameter as well. With a short filament, it was not possible to observe a flapping state, but it kept straight. On the other hand, a larger filament can present the bistable state, which means rest or flapping state that depends on the initial conditions.

A year later a Navier-Stokes and Eulerian-Lagrangian formulation was published. Zhu and Peskin, Farnell [37] presented a work which explained the behaviour of several stable modes; probing the viability of the changing state between an oscillatory mode into a non-oscillatory mode by varying the length of the filament. It contradicted other experimental works and proposed that the creation of the vortices begin at the leading edge of the filament. According to Farnell, vortices grow in filament curvature during their travel downstream and their release from the trailing edge of the filament.

In spite of the numerical simulations presented by Zhu and Peskin or Farnell that have described the flag behaviour, in 2005 Shelley [82] presented an experimental work of a heavy, streamlined and elastic body interacting with a fluid. In his work, Shelley found that above a critical velocity, the body begins to flap with a Strouhal frequency similar to animal locomotion. As well as Shelley, Argentina [14] tried to explain the transition between the flag remaining parallel to the flow and when it begins to flap. Argentina’s study was able to predict a critical speed for the onset of flapping as well as the frequency of flapping.

By 2006, Schouveiler and Eloy [35] modelled the flutter of a rectangular cantilever plate in an axial flow. In this approach, several values of aspect ratio and two different boundary conditions were tested, clamped and pinned-free plate. Similar to Farnell, they tried to explain the cause of instabilities [32]. Schouveiler and Eloy explained that instabilities came from a competition between stabilising fluid pressure and the stabilising flexural rigidity of the plate. Schouveiler and Eloy also demonstrate that it is possible to show that a finite plate is more stable than an infinite one. The model presented by Schouveiler and Eloy was used to study the optimal swimming [34] and [31], the vortex pattern generated by a heavy flat plate [74], the kinematics of cilia [33] and drag reduction [62].

At the same time, Connell [27] developed an outstanding study of flexible filaments classifying its behaviour. In his work, Connell presented an (FSDS) fluid structure direct simulation of a thin and flexible flag for high extensional rigidity and bending rigidity. This study was one of the first in the controlled movement of the filament by three non-dimensional parameters: Reynolds
number, mass ratio and bending stiffness. In this work, three different states as functions of mass ratio were discovered. The first of these is a stable state, the second one shows a periodic limit cycle, which increases with the increment of the mass ratio, the third is a chaotic state related to a larger mass ratio. It was observed that for the largest mass ratios the distribution was way of strong vortex couple from the centreline with rapid changes in tension and dynamic buckling.

It wasn’t until 2007 when Huang [51] simulated the behaviour of a flexible flag. He used his improved version of the immersed boundary method, [49] and[50], for simulating flexible filaments based on the Navier-Stokes solver. In this work, the fractional step method and a staggered Cartesian grid system were used. Similar to previous works, the movement of the filament was modelled on a Eulerian grid, while the inextensible immersed body does it through a Lagrangian grid and both interact via the feedback law. The work studied the small vortex created by the flapping filament. Huang observed the bistable state modifying the length of the filament and the effects of boundary conditions at a fixed end (supported and clamped). In Huang’s previous publications, during 2014, on a single filament, he developed a simulation of a flexible flag immersed in a uniform flow. It included the relation between parameters as a mass ratio, bending rigidity and a Reynolds number. Furthermore, he found three different dynamical states: stretched straight, regular flapping state and irregular flapping state. His model was compared to previous studies; for example Alben, Eloy, Connell or Michelin.

In 2008 Michelin [67] showed a two-dimensional model able to describe the long-term behaviour of a finite and flapping flag. This model was able to reproduce the rest and characteristic flapping states successfully. The elastic strip was considered inextensible and infinitely thin. In this work, three different states were observed for a given inertia ratio. If a low wind speed $U^*$ increases above the critical value, the position became unstable. For intermediate values, a periodic flapping appeared after a transient regime, once velocity increases and the periodicity is broken down. Michelin [68] also published a document where frequency domain is analysed along with the performance of vortex shedding produced by a flexible flag.

A new alternative, proposed to tackle the interaction of flapping bodies immersed in a uniform flow, was presented in 2013 by Favier [39]. Like Zhu and Peskin’s work, Favier defined two meshes, a Eulerian one for a fluid and a Lagrangian one for an immersed body. The main difference of Favier’s work is the use of Lattice Boltzmann equations instead of the Navier-Stokes to avoid pressure correction and make an improved version. Also, Favier defined the immersed body with a set of markers and updated the position of the centre of mass and its rotation through Newton’s laws and the conservation of angular momentum $T$.

Alben [12] recently showed a numerical simulation of a flag flutter in a channel flow. In the case of the heaviest flags, greater confinement increased the size of the instability region, while in the case of lighter flag confinement did not have a significant influence. In this work, the multiple flapping states were found. If channel walls approached the flag, the flapping amplitude decreased in proportion to the near wall distance.
CHAPTER 2. LITERATURE REVIEW

We have already commented about some of the different experiments used to understand the behaviour of a flexible flag immersed in a uniform flow. It is important to mention that most of this paper has focused its efforts on describing the conditions and limits under which the flexible flag modifies its behaviour. In the next section, we are going to summarise some of the relevant works on arrays of flags.

2.3 Arrays of flapping and bending bodies

In previous sections, we have already talked about the studies for a single filament. However, researchers and engineers have also tried to explain the interaction of several flexible bodies immersed in uniform flow and how they perform under different configurations.

In literature, it is possible to find several possible arrays, side by side; or for example, a leader flag etcetera. Figure 2.4 presents an example of an array of flapping bodies with a lead body followed by two flags side by side.

![Figure 2.4: Configurations of flapping bodies immersed in an axial flow.](image)

Based on previous work, one of the first studies was made during 2003 by Zhu and Peskin when they developed a numerical simulation of two flexible filaments [97]. In this model two different modes of sustained oscillation were found, the first one is called parallel flapping and the second one is called mirror-image clapping, and this performance depends on the separation between filaments. The experiments agreed qualitatively with Zhang’s experiments, and complementary to his work, Farnell [36] studied two side by side filaments. Farnell found that filaments can oscillate in different phases: in-phase and in anti-phase. Farnell provided information on how the interaction changes when the separation between them is modified. For small distances, Farnell identified a fundamental frequency that produces an asymmetric system, while for large separations the system exhibits symmetric motion. Huang used a similar configuration [49]. He located two filaments side by side and analysed the vortices shedding from both filaments. Huang confirmed that at the critical distance the filaments flap out-of-phase, but for longer or smaller distances both filaments oscillate in-phase.

In 2005, Jia [54] presented a theoretical and experimental work. Until then most of the experiments were focused on filaments with the same length, but there were few experiments where the filaments have different lengths. In the case of filaments with the same length, Jia identified four different modes: the stretched-straight mode, the anti-symmetrical in-phase mode,
the symmetrical out-of-phase mode and the indefinite mode. Theoretically, these four modes were defined in the case of two filaments of different lengths.

Most of the works presented before 2009 considered just two flexible flags side by side, however, Michelin [67] developed a model able to deal with N flexible plates immersed in a uniform flow. Michelin was interested in analysing the effects of the number of plates and the distance between them. In the case of two plates, Michelin observed the in-phase and out-of-phase modes. In the situation of three plates, Michelin finds three different states: symmetric, in-phase and out-of-phase.

On the other hand, by 2007 Tang [85] used numerical simulation to study the dynamics of two parallel cantilever flexible plates immersed in an axial flow. In his model, Tang used the in-extensibility condition and an unsteady lumped vortex model to analyse the dynamics of the system, the instability and the post-critical behaviour. They found that the flutter threshold is a function of the separation between the two plates and also that the two plates may oscillate out-of-phase and in-phase modes.

The study of vortices shedding inspired the work developed by Kim and Huang [57] in 2010. They designed a study where two tandem flexible flags interact with each other between $200 \leq Re \leq 400$. In this case, they studied the vortices and its destructive and constructive interaction, where the constructive mode increased drag on the downstream flag, while for a destructive case it is possible to observe a decrease in drag on the downstream flag. From this study, it was concluded that the drag reduction for the downstream flag could be optimised for parameters as Reynolds number, streamwise and spanwise gap distances and the bending coefficient of the flexible flag. Huang also participated in the work developed by Uddin [87]. In this work Uddin presented the numerical simulation of an array of flexible filaments, inspired by fish schooling, to study the interaction among flexible flags. In this paper the focus mainly on the interaction between body-vortex and vortex-vortex for different configurations; triangular, diamond and canonical. This paper reports that drag coefficient for downstream bodies decreases if it is adjusted streamwise and spanwise distance and bending. It is important to mention that for single frequency and multi-frequency they correspond with the construction and destruction of the vortex and this performance explains drag reduction in downstream flags.

In 2014 a similar approach as presented to his previous work, Favier [39] considered two filaments interacting side by side at $Re=300$ with an initial angle of $\theta - 18$ degrees. In this work, they found that when spacing is 0.1, the two filaments present a regular flapping. When the distance is equal to 0.3 asymmetrical out-of-phase oscillation appears after the transient period; if spacing increases a little more, the filaments present different movements but the wakes generated by the filaments interact with each other. In the case of a more significant distance, the filaments move harmonically, but the wakes do not mix. In this study, the performance of three filaments was studied. In the case of small spacing, the filaments move symmetrically, and they present just one wake, while if the spacing is increased it is possible to see that the inner
## Table 2.3: Array of flexible or bending bodies.

<table>
<thead>
<tr>
<th>Author</th>
<th>Parameters</th>
<th>Configuration</th>
<th>Measurements</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zhu (S)(2003)</td>
<td>Re=200, d/L=0.1-0.6, M</td>
<td>2 filaments, Different distances</td>
<td>Tail position, vorticity contours</td>
<td>Distinct modes of sustained synchronized flapping</td>
</tr>
<tr>
<td>Farnell(S) (2003) [36]</td>
<td>Re=500, d/L=0.1-0.5, K=0.013</td>
<td>2 filaments axial flow</td>
<td>Frequency, y displacement</td>
<td>In-phase (asymmetry) or in anti-phase (symmetry) depending on separation</td>
</tr>
<tr>
<td>Huang(S) (2007)</td>
<td>Re=300,0.21&lt;d/L &amp; d/L &gt;0.21, M=1.5, y=0.001</td>
<td>2 filaments same length, different inter-distances</td>
<td>Instantaneous vorticity contours</td>
<td>In-phase and out-of-phase flapping observed at d = 0.1 and 0.3</td>
</tr>
<tr>
<td>Jia (T&amp;E)(2007)</td>
<td>Re=2000, d/L=0.8-18.5</td>
<td>2 filaments same &amp; different lengths</td>
<td>Coupling modes, frequency</td>
<td>3 coupling modes for different lengths</td>
</tr>
<tr>
<td>Michelini(S)(2009)</td>
<td>L, B, M</td>
<td>N flexible filaments, uniform flow</td>
<td>Couple motion of N plates</td>
<td>Linear Stability analysis (2 in &amp; out phase), (3 symmetric, 2 in &amp; out phase)</td>
</tr>
<tr>
<td>Tang (S) (2009)</td>
<td>Re=0.2, k=1.0x10^6, d/L</td>
<td>2 cantilever flexible plates, axial flow</td>
<td>Trajectory, frequency spectra</td>
<td>In-phase &amp; out-of-phase analysis</td>
</tr>
<tr>
<td>Kim (S) (2010)</td>
<td>200&lt;Re&lt;400,0.1&lt;cG&lt;2.0, 0.45cG&lt;0.45, 0.001&lt;k</td>
<td>2 tandem flexible flags, axial flow</td>
<td>Cd, y displacement, vorticity,</td>
<td>Constructive mode increased the drag &amp; destructive mode decreased the drag</td>
</tr>
<tr>
<td>Yuan (S) (2013)</td>
<td>Re=0.3-0.6, K=1.8e-3, 3-2.0e-3, L=0.5-1.1, d=0.1-1.0</td>
<td>Filament in wake of a cylinder, 1 &amp; 2 filaments side by side</td>
<td>Cd, Cl, y displacement, tail trajectory, Vorticity</td>
<td>Efficient mode to coupled momentum exchange and IBM</td>
</tr>
<tr>
<td>Udding(S) (2013)</td>
<td>Re=200, M=1.5, d/L, y=0.22</td>
<td>Triangular array of 3 or 4 filaments</td>
<td>Cd, y displacement, power spectra, vorticity</td>
<td>Downstream flags synchronize vortex shed from the upstream flag</td>
</tr>
<tr>
<td>Favier (S) (2014)</td>
<td>Re=200, d/L=0.021</td>
<td>2 &amp; 3 filaments, Different distances</td>
<td>Tail position, y displacement</td>
<td>In-phase (M1), out-of-phase (M2), in-phase with &amp; out of phase flapping (M3)</td>
</tr>
</tbody>
</table>

 filament is affected by the outer filament and almost keeps quasi-stationary. On the other hand, when spacing is big enough, it is possible to see that they do not interact with each other and their wake are independent.

We have already reviewed some of the relevant papers for the arrays of filaments; we have identified the configuration, and we have identified coincidences and differences. In the next section, we will summarise different experiments where a flexible body is attached to a bluff body.

### 2.4 Bluff body attached to a flexible filament

Like the studies of single and arrays of flexible flags, several experimental, mathematical and numerical simulations have been published to identify the effects in downstream flow. All these efforts focused on identifying engineering applications. This section presents a summary of the
2.4. BLUFF BODY ATTACHED TO A FLEXIBLE FILAMENT

<table>
<thead>
<tr>
<th>Author</th>
<th>Parameters</th>
<th>Configuration</th>
<th>Measurements</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Favier [38]</td>
<td>Re=200</td>
<td>Coated circular cylinder</td>
<td>Cd,Cl,frequency,vorticity</td>
<td>Coating capable of increase global aerodynamics performance</td>
</tr>
<tr>
<td>Jia (E)(2009)[55]</td>
<td>Re=300,L=5-30</td>
<td>Filament behind a bluff body</td>
<td>Tail position,frequency, vorticity</td>
<td></td>
</tr>
<tr>
<td>Niu (E)(2011)[70]</td>
<td>Re=200,ρhairs=985.73 kg/m3,EI=3.4x10^4 N/m^2, L=0.5-0.105, N</td>
<td>Coating disk</td>
<td>Cd</td>
<td>synchronization of vortex shedding depends of spacing,length,coating areas</td>
</tr>
<tr>
<td>Bruker (E) (2012) [? ]</td>
<td>L=0.05c,0.1 and 0.2c, Re = 77x10^3</td>
<td>Hairy flaps located at lower part on the wing</td>
<td>Cd, Vorticity</td>
<td>slender flaps are able to delay stall by a factor of 2-4</td>
</tr>
<tr>
<td>Kunze [59]</td>
<td>5000&lt;Re&lt;31000</td>
<td>Coated circular cylinder</td>
<td>Energy, velocity,vorticity</td>
<td>increased of vortex shedding frequency is related to drag reduction</td>
</tr>
<tr>
<td>Venkataraman2012 [88]</td>
<td>Re=1100</td>
<td>Coated airfoil</td>
<td>Cd,Cl, Pressure</td>
<td>Modification of vortices improve aerodynamics</td>
</tr>
<tr>
<td>Bagheri (S)(2012)[17]</td>
<td>Re=100,ρratio=0.005-0.1,Le=0-1.5, Ky</td>
<td>Flapping body behind a circular cylinder</td>
<td>Cd,Cl,y displacement,Vortex Shedding frequency</td>
<td>flapping body influences symmetry vortex shedding</td>
</tr>
</tbody>
</table>

Table 2.4: Array of flexible or bending bodies.

most relevant publications.

In 2009 Favier presented [38] a numerical simulation of the flow around a circular cylinder partially coated with hair, to investigate the capability of the coating to adapt to the surrounding flow at Re=200 and to discover the benefits of the flow control mechanism. Favier found a set of parameters corresponding to a realistic coating (length of the elements, porosity, rigidity), yielding an average drag reduction of 15 per cent and a decrease of lift fluctuations by about 40 per cent, associated with a stabilisation of the wake.

Another configuration was that presented by Jia [55]. Jia reported on a flexible filament located in the wake of a circular cylinder. A flexible filament is forced to vibrate by the periodic vortices shedding from an obstacle. Jia identified three different modes depending on the distance between filament and cylinders. Jia relates his research with to fish locomotion; he explained how fish need to control their head position with vortex and an increase in the wavelength of their body undulation. Jia also commented that the performance of the flexible filaments reveals how organisms take advantage of their elasticity to interact with the periodic flow to obtain energy and produce thrust.

Jia and Favier were not the only researchers that worked with cylinders, Niu [70] experimented with a hairy disk immersed in a soap film. In his experiments, Niu reported a 17 per cent drag reduction if it the proper length filament, density and coating area are selected. Niu commented that the primary mechanism of drag reduction is the bending, adhesion, and reinforcement of hairs trailing the disk.

Bagheri [17] also worked with cylinders. In his experiments, Bagheri considers a two-dimensional inextensible elastic filament with a length $L_s$ and flexural rigidity $B$ attached
to a 2D rigid circular cylinder of diameter $D$. Bagheri calculated the fluid and solid motion through momentum and conservation equations. He identified three simulations with long and short filaments with small rigidity: larger filaments presented sinuous propagation that is amplified at the free end. Along with Von Karman, he also identified the synchronisation of motion. The asymmetric presented a reduction in drag. For the long filament, the vortex was kept unaltered. On the other hand, short filaments induced a compressive fluid force, which promotes vorticity due to the filament resistance to be compressed.

Until this point, most published documents studied the interaction between flag or filaments and circular cylinders, but it wasn’t until 2012 that Brücker [22] studied the interaction of near-wall turbulence with hairy surfaces in a turbulent boundary layer flow along a flat plate in an oil channel at $Re = 1.2 \times 10^6$. This study showed that, as a consequence, the hairs aligned with the stream-wise direction and near-wall high-frequency disturbances, excited by the passage of turbulent sweeps, were dampened over their course along the carpet. Thus, the study concludes that hairy surfaces may be of benefit for turbulent drag reduction. For a NACA profile, Brücker [23] extend his study to cover a set of self-adaptive hairy flaps. The hairy flaps were located on the lower part of the wing and with chord-length of the flow around a NACA0020 airfoil of Reynolds number flow ($Re=77 \times 10^3$). The motion of the flaps and the flow field were measured simultaneously at the high temporal resolution with high-speed PIV (Particle Image Velocimetry). Bruker’s results showed that with a critical value it is possible to demonstrate that in some configurations the slender flaps are indeed able to delay stall by a factor of 2-4 when compared with a clean airfoil. The correlation between flap motion and velocity distribution showed that the hairy flaps indeed prevent backflow induced by vortex structures.

Brücke wasn’t the only one studying a coated airfoil. Venkataraman [88] presented a numerical simulation at low Reynolds of a passive actuation technique that entails covering the suction side of an airfoil with a poroelastic carpet. A parametric analysis showed that such a coating could affect the topology of the flow in the proximity of the rear of the airfoil, by adapting spontaneously to the separated flow. Venkataraman suggested that by selecting suitable characteristics for this carpet, it is possible to reduce drag and increase lift. This phenomenon modifies the scales length of the shed vortices.

Among the experiments proposed by Brücke, we can find the introduced with Kunze [59]. This experiment presented a set of PIV experiments to test flexible and self-adaptive hairy-flaps. Their experiments used a circular cylinder, with and without hairy flaps at $5000 < Re < 31000$. The analyses showed that the hairy-flaps alter the natural vortex separation cycle in such a way that the vortices do not shed in a zig-zag like an arrangement as in the classical von Kármán vortex street, but in a row in line with the cylinder wake axis. Thus, the wake-deficit reduces. Furthermore, flow fluctuations were considerably reduced; by about 42 per cent in streamwise and 35 per cent in transversal direction compared to the reference case without hairy-flaps. The condition for this mode change was the lock-in of the vortex shedding with a wave running through
the flexible hair bundles in a transversal direction at the aft-part of the cylinder. Consequently, the vortex shedding frequency increased, and both, the length of the separation bubble and the drag force decreased.

2.5 Summary of chapter 2

- Splitter plates are an alternative to reduce drag and suppress vortex shedding.
- Rotating splitter plates present a vital reduction of drag depending on the angle of rotation and splitter plate length.
- Flapping bodies behind a bluff body can be used to control wake.
- Most documentation reviewed the behaviour of the flag immersed in a uniform flow such parameters as mass ratio, rigidity and Reynolds number.
- There are several possible approaches to investigate this phenomenon: experimental, theoretical and simulations. In the case of simulations most of the studies reported used the Immersed Boundary Method with either the Navier-Stokes or the Lattice Boltzmann Method.
- Papers focus their attention on the understanding the behaviour of the flag. They measure frequency, amplitude, tail trajectory, vortex shedding as well as the flapping pattern.
- In the case of arrays of flags, there are several possible arrays. In these papers, we see that researchers have identified several possible modes according to the distance between them. In-phase method vortices are shed at the same frequency, while out-of-phase shed at a different rate.
- For experiments with a bluff body used circular cylinders.
- Most of the experiments were done at low Re.
In this chapter, we will give a description of Lattice Boltzmann methodology (LBM), and we will cover the kinetic theory foundations and which clarify the general framework. Furthermore, we will describe the implementation and details of the LBM algorithm. In the first section, we will talk about numerical simulation alternatives for fluid flow simulation, based on the Navier-Stokes or the Lattice Boltzmann. Later, we will discuss Ludwig Eduard Boltzmann’s work and the historical context under which his kinetic theory was developed. This chapter will also include a description of algorithm implementation, boundary conditions and the process to convert among non-dimensional units and lattice units.

3.1 Introduction

Computational fluid dynamics (CFD) is a set of methodologies able to perform the behaviour of fluid flow through a computer. To simulate fluid flow, we solve the laws that govern the movement of the fluids and the boundaries of the system. The main idea under CFD is to transform a system in a virtual environment, where imitate experimental investigation but visualising whole system behaviour with high levels of accuracy.

Computational fluid dynamics divides into five different steps:

- Step 1: Select the mathematical model, in other words, define the level of approximation.
- Step 2a: Space discretization (grid generation)
- Step 2b: Discretization of the equations.
- Step 3: Analyzed numerical scheme, its properties as of stability and accuracy.
- Step 4: Solution of numerical scheme
• Step 5: Graphic post-processing of data solution.

Computational fluid dynamic, or CFD, is associated with the solution proposed by the Navier-Stokes equations. The Navier−Stokes equations were developed independently in the early 1800's by Henri Navier in France and George Stokes in England. However, the Lattice Boltzmann method (LBM) has emerged as an alternative to simulate fluid flows, however before to describing LBM methodology it is important to summarise other methods to justify the decision to use statistical mechanics instead of continuum mechanics.

In the next sections, we are going to explain the Navier-Stokes equations as well as Lattice Boltzmann Methodology.

3.2 The Navier Stokes equations

The central mathematical description for all theoretical fluid dynamics models is given by the Navier-Stokes equations, which describe the motion of viscous fluid domains.

The mathematical models of fluid motion have emerged since the end of 19th century after the industrial revolution. Sir Isaac Newton was the first in describing the viscous fluid motion in his paper "Principia" (1687), where he investigated the fluid behaviour under constant viscosity. Later, the equations of inviscid flow were developed by Daniel Bernoulli (1738) and Leonhard Euler (1755), and they are known as Euler’s inviscid equations. Claude-Louis Navier presented other studies in 1827, Augustin-Louis Cauchy in 1828, Siméon Denis Poisson in 1829, and Adhémar St.Venant in 1843 but it was until 1845 when Sir George Stokes derived the equation of motion of a viscous flow by adding Newtonian viscous terms. The final form of this equations was called the Navier-Stokes equations.

The Navier-Stokes equation consists of a time dependant continuity equation for the conservation of mass, which can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \quad (3.1)$$

three time dependent conservation momentum equations:

$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} = -\frac{\partial P}{\partial x} + \frac{1}{Re} \left( \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right) \quad (3.2)$$

$$\frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho u v)}{\partial x} + \frac{\partial (\rho v^2)}{\partial y} + \frac{\partial (\rho vw)}{\partial z} = -\frac{\partial P}{\partial y} + \frac{1}{Re} \left( \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right) \quad (3.3)$$

$$\frac{\partial (\rho w)}{\partial t} + \frac{\partial (\rho u w)}{\partial x} + \frac{\partial (\rho v w)}{\partial y} + \frac{\partial (\rho w^2)}{\partial z} = -\frac{\partial P}{\partial z} + \frac{1}{Re} \left( \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right) \quad (3.4)$$

These equations have three independent spatial coordinates $x,y,z$, a time variable $t$ and dependant variables such as pressure $P$, density $\rho$ and three components of velocity vectors $u_x,u_y$ and $u_z$. 

38
The terms on the left-hand side of the momentum equations are called the convection terms of the equations. Convection is a physical process that occurs in the flow when the ordered motion of the flow transports some property. The terms on the right-hand side of the momentum equations, which are multiplied by the inverse Reynolds number are called the diffusion terms. Diffusion is a physical process that occurs in flow when the random motion of the molecules transports some property. Diffusion is related to the stress tensor and the viscosity of the fluid.

As previously mentioned, the Navier-Stokes equations are the basis of CFD; however, the modelling of turbulence, which is the result of diffusion in the flow is the main factor to model the behaviour of the fluid. A turbulent flow is characterised by a range of irregular and chaotic eddies of different sizes. The largest eddies have the similar characteristic length to the domain and, obtain energy from mean flow. This energy is passed down gradually from the largest eddies to the smallest ones until all the energy is dissipated by viscosity. This process is known as a turbulent scale cascade.

Different scales present in a turbulent flow can be selected. There are many different approaches presented (various scales) that can be classified into three categories: Direct numerical simulation, Large-eddy simulations and Reynolds-Averaged Navier Stokes.

3.2.1 Navier-Stokes solvers:

As we mentioned before, CFD [2] has become an essential tool for design engineers because they can verify products according to specifications in the very early stages of the design process. This has accelerated the process of product development considerably. The use of CFD in engineering has been used in many areas, figure 3.1 presents a summary of the sectors where it has been applied.

![Figure 3.1: CFD Sectors.](image)

Due to CFD has emerged as a tool for the engineering process, there have been efforts by companies and researchers to provide different alternatives. Commercial CFD codes require a license while open source codes do not need them. In both cases, it is necessary to solve the
CHAPTER 3. LATTICE BOLTZMANN METHOD

<table>
<thead>
<tr>
<th>Feature</th>
<th>Open-Source</th>
<th>Commercial</th>
</tr>
</thead>
<tbody>
<tr>
<td>Customization of physical models</td>
<td>Excellent</td>
<td>Good</td>
</tr>
<tr>
<td>Customization of numerical schemes</td>
<td>Excellent</td>
<td>Poor</td>
</tr>
<tr>
<td>Authenticity of physical models</td>
<td>Excellent</td>
<td>Good</td>
</tr>
<tr>
<td>User friendliness</td>
<td>Poor</td>
<td>Excellent</td>
</tr>
<tr>
<td>Free Technical support</td>
<td>Poor</td>
<td>Good</td>
</tr>
<tr>
<td>Paid technical support</td>
<td>Excellent</td>
<td>Excellent</td>
</tr>
<tr>
<td>Academic use</td>
<td>Excellent</td>
<td>Good</td>
</tr>
<tr>
<td>Industrial use</td>
<td>Poor</td>
<td>Excellent</td>
</tr>
</tbody>
</table>

Table 3.1: Comparative between open source and commercial CFD software.

Navier-Stokes fluid flow equations. Table 3.1 shows a comparative among some features to understand the advantages and disadvantages of these alternatives.

In table 3.1 is possible to observe that commercial CFD software, in general, presents an excellent user-friendly interface, technical support and they have widely used it in industry, however in most of the cases it is not possible to customise numerical schemes. On the other hand, open source codes permit the customisation of numerical schemes or physical models, but in most of them, there is not present a user-friendly interface. In both cases, there is excellent support, but commercial CFD has more significant support, due to they have an official web page with tutorials and examples, while open source codes depend more than open source community on explaining their use.
3.3 Lattice Boltzmann solvers

In previous sections, we described solvers based on the Navier-Stokes equations; now it is time to describe solvers based on the Lattice Boltzmann method (LBM). However before to explain the theory and methodology, it is necessary to remark the importance that this methodology has earned over the years.

During the last decades, CFD based on lattice Boltzmann method has become more popular. By 1980 just a pair of papers using this methodology were published while for early 2000 more than 250, figure 3.2. These papers were published in more diverse areas like physics, engineering, mathematics, computer sciences, chemical engineering etc. Figure 3.3 shows classification by the topic of all papers published since 1980.

The increase of this methodology has propitiated that many companies or researchers create
Table 3.2: CFD solvers available with Lattice Boltzmann methodology simulations.

<table>
<thead>
<tr>
<th>Name</th>
<th>O.Source</th>
<th>Solver</th>
<th>Customization</th>
<th>Interface</th>
<th>Parallel Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exa PowerFlow</td>
<td>★</td>
<td>BGK,MRT</td>
<td>★</td>
<td>✓</td>
<td>CUDA</td>
</tr>
<tr>
<td>Xflow</td>
<td>★</td>
<td>BGK,MRT</td>
<td>★</td>
<td>✓</td>
<td>CUDA/OpenCL/MPI</td>
</tr>
<tr>
<td>Numeca</td>
<td>★</td>
<td>BGK,MRT</td>
<td>★</td>
<td>✓</td>
<td>CUDA/OpenCL/OpenMP/MPI</td>
</tr>
<tr>
<td>Sailfish</td>
<td>✓</td>
<td>BGK,MRT,ELBM</td>
<td>✓</td>
<td>★</td>
<td>CUDA/OpenCL</td>
</tr>
<tr>
<td>OpenLB</td>
<td>✓</td>
<td>BGK,MRT</td>
<td>✓</td>
<td>★</td>
<td>MPI</td>
</tr>
<tr>
<td>LUMA</td>
<td>✓</td>
<td>BGK, MRT</td>
<td>✓</td>
<td>★</td>
<td>MPI</td>
</tr>
<tr>
<td>Palabos</td>
<td>✓</td>
<td>BGK, MRT</td>
<td>✓</td>
<td>★</td>
<td>MPI</td>
</tr>
</tbody>
</table>

their own LBM code. Some of the codes available in the market as Exa Power flow [1],Xflow [9], Numeca [5], Sailfish [8], OpenLB [6] and Palabos [7]. With the intention to identify their capabilities and characteristics. Table 3.2 shows a summary of the most frequently used LBM codes, as well as the implementation developed by the University of Manchester.

Similar to the conventional Navier-Stokes CFD commercial codes, CFD codes based in Lattice Boltzmann methodology present friendly interfaces, support services, a set of conferences and forums as well as teaching services that improve the user experience. Most of these codes also present different strategies of parallelisation, in most cases optimised for Nvidia hardware.

In the case of LBM open source codes, it is difficult to find a large community of users or teaching services that help new users with the adoption. However, the most significant advantage is the possibility to have access to source code, which offers the user the chance to customise it, as well as participate in its development. Regarding the parallelisation aspect, the most significant advantage is that the user does not need a specific brand of hardware to use it.

Up until now, we have presented an introduction to some of the LBM codes available in the market and shown how it has increased their use in recent years, however, it is time to introduce its basis as well as explain its implementation.

3.4 Boltzmann’s background and kinetic theory

Ludwig Boltzmann was a physicist born in Austria in 1844. He is known for his work in statistical mechanics. In his work based on atomic theory, Boltzmann explained the second law of thermodynamics and demonstrated that it is possible to blend mechanics laws with the theory of probability. Boltzmann’s contributions are not just in the field of statistical mechanics, electromagnetism theory and black body radiation based on Stefan’s law, but also, and more importantly for this work in the area of the calculation of the kinetic theory of gases.

The kinetic theory developed by Boltzmann explains the dynamics of non-equilibrium processes and their relaxation to thermodynamic equilibrium. Based on the hypothesis that the matter is composed of a large and finite number of molecules travelling in constant random motion, caused by their collision, it is possible to explain the behaviour of the macroscopic properties of fluids as pressure, temperature, viscosity or thermal conductivity based on this motion.

Boltzmann developed his theory based on the following assumptions:
3.5. MAXWELL BOLTZMANN DISTRIBUTION

Figure 3.4: Ludwig Boltzmann

- The smallest part of the fluid is known as a molecule.
- Molecules must have the same mass.
- The total number of molecules should be large enough to be applied in statistical treatment.
- All molecules constantly move, randomly and rapidly.
- Molecules are perfectly spherical, and they collide elastically among them perfectly.
- The interaction among molecules is just during collisions, relativistic or quantum mechanical effects are negligible.
- The kinetic energy of fluid particles depends on the absolute temperature of the system.
- It is possible to neglect the time collision between molecules and the container, due to the time between successive collisions being a lot higher.
- Due to molecules having mass, molecules are affected by gravity.

The mesoscopic kinetic theory developed by Boltzmann is used to describe the distribution of particles of a fluid, in other words, the behaviour of any fluid.

In the next section, we are going to explain the Lattice Boltzmann methodology.

3.5 Maxwell Boltzmann distribution

Boltzmann was the first to generalise the Maxwell distribution of any large system. According to him, there is a deep connection between entropy and the statistical analysis of possible states of a large system. If a system increases its entropy, macroscopic variables will modify their corresponding values.

Boltzmann established that for any system, large or small, in thermal equilibrium, at a given temperature $T$, the probability of being in a particular state at energy $E$ is proportional to $e^{-E/kT}$
CHAPTER 3. LATTICE BOLTZMANN METHOD

\[ f(E) = A e^{-E/kT} \]  \hspace{1cm} (3.5)

If \( E \) is the \( x \)-direction kinetic energy:

\[ E = \frac{1}{2} m c_x^2 \]  \hspace{1cm} (3.6)

For a normalized probability function, the probability function integrated for all values of velocity (from minus to plus infinity) should be one:

\[ \int_{-\infty}^{\infty} A e^{\frac{m c_x^2}{2kT}} = 1 \]  \hspace{1cm} (3.7)

Therefore:

\[ A = \sqrt{\frac{m}{2\pi kT}} \]  \hspace{1cm} (3.8)

The probability of finding velocity \( c_x \) is:

\[ f(c_x) = \sqrt{\frac{m}{2\pi kT}} e^{\frac{mc_x^2}{2kT}} \]  \hspace{1cm} (3.9)

Considering the probability of a three dimensional velocity \( c \) where:

\[ c^2 = c_x^2 + c_y^2 + c_z^2 \]  \hspace{1cm} (3.10)

The probability of \( f(c) \) is the probability multiplication of each function:

\[ f(c) = f(c_x)f(c_y)f(c_z) \]  \hspace{1cm} (3.11)

Which leads to:

\[ f(c) = \left[ \frac{m}{2\pi kT} \right]^{3/2} e^{-\frac{mc^2}{2kT}} \]  \hspace{1cm} (3.12)

Combining the equation 3.11 and 3.12

\[ f(c) = \left[ \frac{m}{2\pi kT} \right]^{3/2} e^{-\frac{mc^2}{2kT}} \]  \hspace{1cm} (3.13)

The last equation is the Boltzmann distribution function, in consequence, the basis of the Lattice Boltzmann Method. In the next sections, we will show its importance as well as the discretisation used for fluid flow simulations.
3.6 Lattice Boltzmann Method

The Boltzmann distribution function presented in previous sections is the basis of the Lattice Boltzmann Methodology. Here, a set of discrete points are going to describe the distribution of particles which move in specified directions. Figure 3.5 shows an example of the different scales: microscopic, mesoscopic and macroscopic scales; section (a) show the aleatory movement ; (b) presents the pre-established directions that particles are permitted to follow and; (c), where the Navier-Stokes equations works, considers the continuum medium.

![Fluid flow scales](image)

Figure 3.5: Fluid flow scales.

The scale, where the Boltzmann theory works, is known as the mesoscopic scale, midway between the microscopic and macroscopic scale. At the microscopic level, particles are moving continuously in space, and they are free to move in any direction, occupy any location in space and travel at any speed. In other words, at this level, it is just about knowing that particles are moving at a specific velocity and phenomenological properties like viscosity, thermal conductivity, temperature and pressure are only the manifestations in the macroscopic world of the particle arrangement in a microscopic world. At a macroscopic scale, which refers to a continuum, quantities such as fluid velocity, density or others previously mentioned, they are tangible, and we can measure them.

In the mesoscopic scale, the distribution of molecules is tracked, and the kinetic theory will define the movement and the distribution of particles in the system. Due to the difficulty in dealing with such a considerable number of molecules, it is possible to deal with them by averaging. The distribution function 3.13 offers the possibility to know the percentage of particles in a specific location with velocities in a certain range at a given time, \( f(r,c,t) \).

In the next section, the physics under Boltzmann’s work will be explained step by step, but we have to keep in mind Boltzmann's distribution function as well as the framework.
3.6.1 Boltzmann Transport equation

The distribution function can statistically explain a system, \( f(r,c,t) \) and it is defined as a number of particles at a specific time \( t \), which are positioned between \( r \) and \( t+dr \) and moves at velocities between \( c+dc \). Consider a system of two particles without any external force, \( F \), defined as \( F=ma \); A collision between them will not take place, this process can be written mathematically:

\[
f(r+cdt,c+Fdt,t+dt)drdc-f(r,c,t)drdc=0 \tag{3.14}
\]

Now, if an external force is applied to a particle, its momentum will increase, and this momentum will transfer to its neighbour. This will be expressed mathematically, \([69]\):

\[
f(r+cdt,c+Fdt,t+dt)drdc-f(r,c,t)drdc=\Omega drdc dt \tag{3.15}
\]

Figure 3.6 shows the before and after when a Force \( F \) is applied.

![Figure 3.6: Position of a particle before and after an external force is applied](image)

The rate of change between the final and initial status is called a collision operator, \( \Omega \). Dividing the equation above by \( drdc dt \) and as the limit \( dt \) tends to 0, the equation can be written as:

\[
\frac{df}{dt} = \Omega(f) \tag{3.16}
\]

The equation above describes the total rate of change of the distribution function, which is equal to the rate of the collision. Since \( f \) is a function of \( r \), \( c \) and \( t \), then the total rate of change can be expanded as:

\[
df = \frac{\partial f}{\partial r} dr + \frac{\partial f}{\partial c} dc + \frac{\partial f}{\partial t} dt \tag{3.17}
\]

Dividing by \( dt \):

\[
df = \frac{\partial f}{\partial r} \frac{dr}{dt} + \frac{\partial f}{\partial c} \frac{dc}{dt} + \frac{df}{dt} \frac{dt}{dt} \tag{3.18}
\]

Where \( a=dc/dt \), the acceleration related to the force \( F \), by Newton's second law, \( F/m \).
3.6. LATTICE BOLTZMANN METHOD

\[
\frac{df}{dt} = \frac{\partial f}{\partial r} c + \frac{\partial f}{\partial c} a + \frac{\partial f}{\partial t}
\]  

(3.19)

Therefore, the Boltzmann transport equation can be written as:

\[
\frac{\partial f}{\partial t} + \frac{\partial f}{\partial r} c + \frac{\partial f}{\partial c} F = \Omega
\]  

(3.20)

The \( \Omega \) is a function of \( f \) that need to be determined to solve the Boltzmann equation. Assuming a system without external force, the Boltzmann equation can be written as:

\[
\frac{\partial f}{\partial t} + c \cdot \nabla f = \Omega
\]  

(3.21)

Note that \( c \) and \( \nabla f \) are vectors and \( \Omega \) is a source term of an advection equation. This equation can be solved along characteristic tangent lines of vector \( c \) if \( \Omega \) is known. However, \( \Omega \) is a function of \( f \) and equation 3.21 is an integro-differential equation which is not easy to solve. In the next section, we will describe the solution developed by Bhatnagar, Gross and Krook.

At this point, we have talked about distribution function \( f \), equation 3.13, and Boltzmann’s transport equation 3.21, but how can we connect them with macroscopic world? The next set of equations show the connection with the macroscopic quantities such as fluid density, \( \rho \), fluid velocity vector \( c \), and the internal energy \( e \):

\[
\rho(r,t) = \int mf(r,c,t)dc
\]  

(3.22)

\[
\rho(r,t)c(r,t) = \int mvf(r,c,t)dc
\]  

(3.23)

\[
\rho(r,t)e(r,t) = \frac{1}{2} \int mu_a^2f(r,c,t)dc
\]  

(3.24)

Where \( m \) is the molecular mass and \( u_a \) the particle velocity relative to the fluid velocity, \( u_a = c - u \) and the equations 3.22, 3.23 and 3.24 are mass, momentum and energy respectively.

3.6.2 Bhatnagar,Gross and Krook (BGK) Approximation

In the last section, we commented on how \( \Omega \) is a source term of an advection of equation 3.21 which is an integro-differential equation not easy to solve. This was until 1954 when Bhatnagar,Gross and Krook [20] obtained a simplified and approximated solution for Boltzmann’s transport equation. This function was called collision operator \( \Omega \):

\[
\Omega = \omega \left( f^{eq} - f \right) = \frac{1}{\tau} \left( f^{eq} - f \right)
\]  

(3.25)

Where \( \omega = \frac{1}{\tau} \) and \( f \) is the equilibrium distribution function. This scheme is called the single time relaxation scheme, due to all nodes relaxing on same time scale \( \tau \), which is the viscous term.
of the Navier-Stokes equation. However, it is possible to obtain higher stability and accuracy through multiple relaxation schemes.

Once equations 3.21 and 3.25 are introduced, Boltzmann’s equation can be written as:

$$\frac{\partial f}{\partial t} + v \cdot \nabla f = \frac{1}{\tau} (f^{eq} - f) \quad (3.26)$$

Discretizing and assuming validity for certain directions, the Boltzmann equation can be written for a specific direction, such as:

$$\frac{\partial f_i}{\partial t} + v_i \cdot \nabla f_i = \frac{1}{\tau} (f^{eq}_i - f_i) \quad (3.27)$$

And completely discretized:

$$f_i(r + v_i \Delta t, t + \Delta t) = f_i(r, t) + \frac{\Delta t}{\tau} \left[ f^{eq}_i(r, t) - f_i(r, t) \right] \quad (3.28)$$

Where the discrete version of the equilibrium function can be defined as:

$$f^{eq}_i = \omega_i \rho \left[ 1 + \frac{e_i \cdot \bar{u}}{C_s^2} + \frac{(e_i \cdot \bar{u})^2}{2C_s^2} \right] \quad (3.29)$$

The equation 3.27 replace the Navier-Stokes equations in comparative with conventional CFD, and it is the most important equation for Lattice Boltzmann methodology. The Navier-Stokes equations and Boltzmann equations are connected, and in the next sections, we will describe this process.

### 3.6.3 Lattice Boltzmann Model

Before explaining the Lattice Boltzmann steps, we have to describe the lattice node behaviour. The Lattice Boltzmann method limits the space lattice as well as space velocities to a set of microscopic velocities $e_i=(e_{ix}, e_{iy}, e_{iz})$. The model can use different types of lattice, cubic or triangular and in the discrete distribution function with or without rest particles.

The most popular way to classify the different lattice methods is the scheme $DnQm$, where $n$ specifies dimensions and $m$ the speeds. There are several possible arrangements for example $D2Q9$, $D3Q15$, $D3Q19$ or $D3Q27$. $D2Q9$ is the most popular scheme and represents a two dimensional model with nine possible neighbours.

The microscopic velocities for a $D2Q9$ are denoted by:

$$\bar{e}_i = cX \begin{cases} (0,0) & i=0, \\ (1,0),(0,1),(-1,0),(0,-1) & i=1,2,3,4 \\ (1,1),(-1,1),(-1,-1),(1,-1) & i=5,6,7,8 \end{cases}$$

Once the simplest element in Lattice Boltzmann methodology is clarified, we can start by saying that the dynamics of the particles inside this unit are idealised. Particles "live" inside
of lattices, each lattice will be associated with a node, each node is associated via a vector with a specific velocity and momentum, these vectors are the lattice velocities, which we mentioned previously.

The interaction of particles in a lattice can be summarized in two main steps: collision and streaming. During the collision step, particles collide locally and are only considered particles at the same node. This step can be written mathematically as:

$$ f_i(x + c e_i, t + \Delta t) - f_i(x, t) $$

The second step is streaming. This step is not local and particles can move according with permitted directions; mathematically we can define this step as:

$$ \frac{[f_i(x, t) - f_i^{eq}(x, t)]}{\tau} $$

Figure 3.8 shows the movement of particles during the collide and stream steps. The interaction among the particles affects the macroscopic hydrodynamics of the model. Since it is a mesoscopic perspective, it is not possible to anticipate completely all of the collisions involved during the process. For this reason, the local collision rules must be able to produce the long-term effects that lead to equilibrium. The simplest way to reach the equilibrium is through the introduction of a collisional operator. In this case, the BGK approach introduced in section 3.6.2.
In the next section, we will explain the implementation of a D2Q9 scheme step by step.

### 3.6.4 Lattice Boltzmann method implementation

Here, we will consider the most common scheme used, which is D2Q9. This refers to nine possible directions during the stream step. These directions refer to microscopic velocities. A LBM code can be calculated as:

- Initialize $\rho$, $\overline{u}$, $f_i$ and $f_i^{eq}$.
- Streaming step: move $f_i \rightarrow f_i^*$ in the direction of $\overline{e}_i$.
- Compute macroscopic \( \rho \) and $\overline{u}$ from $f_i^*$ with equations 3.22 and 3.23.
- Compute $f_i^{eq}$.
- Collision step: calculate the updated distribution function $f_i = f_i^* \cdot \frac{1}{\tau} (f_i^* - f_i^{eq})$
- Repeat steps from Stream until Collision step.

![Flowchart of the most fundamental steps in a LBM implementation.](image)

3.7 Hydrodynamic forces over solid objects and complex geometries

In this section, we are going to describe a set of boundary conditions that will help us to define the interaction between fluid force and solid objects. Firstly, we are going to represent the different methods used to deal with immersed bodies and secondly, we are going to explain their implementation in Lattice Boltzmann Methodology.
3.7. HYDRODYNAMIC FORCES OVER SOLID OBJECTS AND COMPLEX GEOMETRIES

3.7.1 Boundary conditions for an immersed body in a uniform flow

Boundary conditions are a set of rules used to manipulated variables through pressure and velocity to give suitable results. The application of boundary conditions is more difficult when the geometry is more complicated. Figure 3.10 presents a summary of 2 approaches used for complex geometries; staircase and non-conform meshes.

![Complex geometries diagram]

The staircase approximation is used when we want to impose non-slip conditions in a regular grid for simple shapes like cylinders. In the following sections, we will describe this approach, and the steps (bounce-back and momentum exchange) needed to calculate force over the immersed body and its implementation on LBM.

In the case of the non-conforming meshes (where makers must be connected, and they are used complex and deformable shapes), we are going to talk about the immersed boundary method and its implementation on LBM.

3.7.2 Staircase approximation

To apply staircase approximation, we need two steps: the first step consists in bounce-back boundary conditions after streaming step and the second one of applying momentum exchange method, both methods are described in the following subsections.
3.7.2.1 Bounce-Back Boundary Conditions

The bounce-back boundary is a condition commonly used to impose the non-slip condition. The main idea is that fluid particles must be bounce-back in the opposite direction (back to the fluid) as soon as they reach any boundary node.

Bounce-back conditions have been studied extensively, and there are several possible implementations; Full way and midway bounce-back are the most common ones. In the case of full way bounce-back, it is the easiest to implement and presents a proper numerical accuracy. Due to the boundary of the fluid is aligned with the lattice points, the incoming points of the distribution functions take the opposite direction once they reach the boundary node.

![Figure 3.11: Full way bounce-back boundary conditions before and after streaming.](image1)

Midway bounce-back uses fictitious nodes located between the boundary wall and fluid nodes. The idea of a mid-way bounce-back is that at a time step $t$, when the distribution function towards the boundary walls, they are going to leave the domain; the collision step is applied to modify the opposite directions of these distribution functions to bounce-back towards the boundary nodes. The distribution functions at the end of these boundary conditions are the post-collision distribution functions.

![Figure 3.12: Midway bounce condition before and after streaming.](image2)

Until this point, we have defined two implementations of bounce-back boundary conditions. In the case of full-way bounce-back, it shows that this solution is first order accuracy due to its
one-sided treatment on streaming at the boundary, while mid-grid bounce-back is the second order of accuracy.

### 3.7.2.2 Momentum exchange method

The model used in this work is developed by Ladd [60] and it is based on a midway bounce-back boundary condition and known Momentum Exchange Method. The model is based on the following assumptions:

- The immersed body is considered as a shell, in both sides of particle boundary the fluid exists.
- The particle at the boundary connect the fluid nodes with the solid nodes.
- The main idea of momentum exchange method is to obtain the difference of distribution function from opposite directions along the boundary links to evaluate the hydrodynamic force acting on the particle.

![Figure 3.13: Scheme of link-base LBM.](image)

Figure 3.13 shows the momentum exchange method, assuming that the moving boundary \( w \) connects solid node \( x_b \) and fluid node \( x_f \) the discrete velocity \( e_i \) will move from \( x_f \) to \( x_b \), a momentum exchange is computed at the boundary velocity to the distribution function which are bounce back from the particle boundary:

\[
f_f(x_f, t + 1) = \tilde{f}_i(x_f, t) - \frac{2\omega_i \rho}{C_s^2} (e_i \cdot u_b) \quad (3.32)
\]

\[
f_i(x_s, t + 1) = \tilde{f}_i(x_b, t) - \frac{2\omega_i \rho}{C_s^2} (e_i \cdot u_b) \quad (3.33)
\]

Where \( C_s \) is the sound of speed and \( u_b \) is the velocity at the intersection. The momentum-exchange is done during streaming step and the momentum exchange value on a fluid-solid link in a time step, known as the force and can be written as:
CHAPTER 3. LATTICE BOLTZMANN METHOD

\[ F_i(x_b, t + \frac{1}{2}) = 2 \left[ \hat{f}_i(x_f, t) - \hat{f}_i(x_b, t) - \frac{2\omega_i \rho}{C_s^2} (e_i \cdot u_b) \right] e_i \]  
(3.34)

\[ F(t) = \sum F(x_b, t) \]  
(3.35)

This calculation is made for each time step and, according to 3.2, is during the application of boundary conditions. In the next section, we will describe the non-conforming meshes and their implementation.

3.7.3 Non-conforming meshes

There are several possible implementations of IBM methodology. In cases where there is a rigid body, it is not necessary to connect neighboring markers, while for deformable bodies, we have to specify these connections. This kind of mesh is known as non-conforming mesh due to it does not have to be aligned with the lattice of the LBM. In the next section, we will describe the process to coupling Immersed Boundary Method and Lattice Boltzmann Methodology.

3.7.3.1 Immersed boundary method

The immersed boundary method was developed in 1970 to study flow patterns around heart valves by Peskin [76]. The methodology is widely used and many other researchers who have developed their versions based on the original; for example Pinnelli [77], Taira [83]. However, most implementations follow a set of similar steps, and we are going to describe them in the following sections, mainly for IB-LBM. First, though, we have to comment that for non-conforming mesh we need two meshes, one Eulerian and one Lagrangian and they describe as:

- A Eulerian grid is defined at the same position as LBM Lattice nodes. It describes as stationary and regular.

- A Lagrangian grid is represented by a set of markers. The function of these markers describes complex geometry and the forces exerted by them will prescribe the velocity at the boundary.

Figure 3.14 represents the Lagrangian markers and Eulerian grid. It is important to clarify that the Lagrangian markers are a set of points that are not part of the Eulerian grid and they can communicate and modify its position in reference with Eulerian grid. In the following section, we will describe how we can couple the dynamic boundary and the fluid. In other words, how fluid knows about deformable body boundaries.
3.7. HYDRODYNAMIC FORCES OVER SOLID OBJECTS AND COMPLEX GEOMETRIES

Figure 3.14: a) Lagrangian markers \( r_j \) b) Eulerian grid \( X \).

3.7.3.2 Coupling Immersed Boundary Method (IBM) and Lattice Boltzmann Method (LBM)

In previous sections, we have presented the discrete version of lattice Boltzmann equation and the discrete version of equilibrium function. In order to combine Lattice Boltzmann equation with body term \( \vec{F} = (F_x, F_y) \) can be expressed as:

\[
f_i(r + v_i \Delta t, t + \Delta t) = f_i(r, t) + \frac{\Delta t}{t} \left[ f_i^{eq}(r, t) - f_i(r, t) \right] + \frac{1}{2} \frac{\omega_i}{C_s^2} \left[ \vec{e} \cdot \vec{F}(\vec{x}, t) \right] + \frac{1}{2} \frac{\omega_i}{C_s^2} \left[ \vec{e} \cdot \vec{F}(\vec{x} + \vec{v}_i \Delta t, t + \Delta t) \right] \Delta t
\]

(3.36)

According to the equation 3.36, Lattice Boltzmann methodology and Immersed boundary method can be separated in three steps, collision, streaming and forcing step.

the collision step:

\[
f'_i(\vec{x}, t) = f_i(\vec{x}, t) - \frac{1}{t} \left( f_i(\vec{x}, t) - f_i^{eq}(\vec{x}, t) \right) + \frac{1}{2} \frac{\omega_i}{C_s^2} \left[ \vec{e} \cdot \vec{F}(\vec{x}, t) \right] \Delta t
\]

(3.37)

the streaming

\[
f''_i(\vec{x} + \vec{v}_i \Delta t, t + \Delta t) = f'_i(\vec{x}, t)
\]

(3.38)

and the forcing step for the moving curved boundary. Collision step:

\[
f_i(\vec{x} + \vec{v}_i \Delta t, t + \Delta t) = f''_i(\vec{x} + \vec{v}_i \Delta t, t + \Delta t) + \frac{1}{2} \frac{\omega_i}{C_s^2} \left[ \vec{e} \cdot \vec{F}(\vec{x} + \vec{v}_i \Delta t, t + \Delta t) \right] \Delta t
\]

(3.39)

As mention before, the curved boundary is defined as a set of Lagrangian markers \( X_k = (x_k, y_k) \) and a force \( \vec{F}_k = (X_{x,k}, Y_{y,k}) \). The Lagrangian makers are distributed at the boundary, the force applied over them must be distributed on neighboring lattice by a delta function. The boundary velocity is obtained through a interpolation velocity from a neighboring lattice. The discretized delta function and the function responding for the interpolation of velocity are both set as the area weighted average function \( \delta h \):

\[
\delta h(\vec{x} - \vec{X}_k) = d_h(x_i - x_k)d_h(y_i - y_k)
\]

(3.40)

where \( d_h \) is the hat function and is defined as:
The lattice spacing is $h$. The most effective range for $\delta h$ is $2 \Delta x$ by $2 \Delta x$ square, where the maximum number of lattice influenced by a Lagrangian boundary marker is four.

The summation of weighted forces over the series of the Lagrangian boundary markers is the total force $\vec{F}(\vec{x})$ and it can be defined as:

$$\vec{F}(\vec{x}) = \sum_k \vec{F}_k \cdot \delta(\vec{x} - \vec{X}_k) \quad (3.41)$$

where the summations over the Lagrangian boundary markers is defined as $\Sigma_k$. In a similar way, the velocity at the Lagrangian boundary marker $\vec{u}(\vec{X}_k)$ is defined as the summation of weighted velocity over four neighboring lattice of the Lagrangian boundary marker $\vec{X}_k$:

$$\vec{u}(\vec{X}_k) = \sum_x \vec{u}(\vec{x}) \cdot \delta(\vec{x} - \vec{X}_k) \quad (3.42)$$

Where $\Sigma_k$ is the summation over the neighbouring lattice of the Lagrangian boundary marker $\vec{X}_k$.

The total force $\vec{F}(\vec{x})$ distributed to a lattice modify its velocity by 3.39 do that the velocity at a Lagrangian boundary marker $\vec{u}(\vec{X}_k)$ obtained by interpolation exactly the prescribed velocity.

### 3.8 Lattice Boltzmann Units

In this section, we will describe the conversion and selection of the lattice units to non-dimensional units. We will identify a set of steps to move among units and also explain the process used in this work to set up the computational domain used in our experiments.

#### 3.8.1 Units conversion and rules to follow

We mentioned in previous chapters that the Lattice Boltzmann Method model is a mesoscopic representation of macroscopic physics, for that reason, it is important to choose the proper units. When we try to select the lattice units, we need to consider two rules:

- There should be an equivalence between the LBM system and the physical system.
- Parameters should be converted properly to get proper accuracy

Usually, when we solve fluid flows, the goal is to address the Navier-Stokes macroscopic equations. However, when we set a mesoscopic simulation, we do not have a straightforward way to pass from the physical $P$ units to LB units. A clear way to move between them is to first
convert to a dimensionless \( D \) system, which is independent of the original physical scale as well as simulation parameters. Just put the correspondence between the physical \( P \) system and the \( LB \) system is made through dimensionless \( D \) numbers or scale independent numbers, 3.43 presents a graphical representation among the three systems, Latt [61].

Physical system (P)  \( \rightleftharpoons \) Dimensionless System(\( D \))  \( \rightleftharpoons \) Discrete System(\( LM \)) \hspace{1cm} (3.43)

Once correspondence among the three systems is clarified, we can define a set of rules to move among these systems:

- Non-dimensional parameters, as Reynolds number \( Re \) are the same for three system.
- The characteristic length \( L_p \) and time scale \( t_p \) are used as transition parameters between physical \( P \) and dimensionless \( D \).
- The discrete space \( \delta x \) and time step \( \delta t \) are used as transition parameters between dimensionless \( D \) and Lattice \( LB \) space.

Once we have clarified these rules, we can say that it is possible to move from Lattice units \( LB \) to physical units \( P \) without passing through dimensionless units \( D \). However, the efficient and accurate process uses dimensionless units \( D \) as a bridge between mesoscopic and macroscopic world.

In the next section, we will present a set of equations that are going to help us to move among different units. Due to our test cases are presented in the non-dimensional system, we will describe the process used to set up an example from non-dimensional system to the lattice system.

### 3.8.2 Non-Dimensional units to Lattice Units

As we mentioned in the previous section, the discrete space interval \( \delta x \) and the reference time \( \delta t \) works as transition parameters between a non-dimensional system and a lattice system and they define as:

\[
\delta x = \frac{1}{N} 
\]

\hspace{1cm} (3.44)

Where \( N \) is defined as the number of lattice points used to discretise the characteristic length, \( \delta t \) can be described as the reference time divided by the number of iterations steps \( N_{iter} \) used to reach this time.

\[
\delta t = \frac{1}{N_{iter}} 
\]

\hspace{1cm} (3.45)
Before to continue, it is important to comment that the Lattice Boltzmann model is quasi-compressible and compressibility could have an impact on numerical accuracy, in order to keep under control simulation, we need to control Mach number in proportion to $u_{0,LB}$, this can be written as:

$$Ma = \frac{u_{0,LB}}{c_s}$$  \hfill (3.46)

Where $c_s$ is the velocity of sound.

Up until this point, we have shown the discrete space interval $\delta x$, the reference time $\delta t$ and taken into account that non-dimensional parameters as Re number remains constant among systems, we can write:

$$Re_{LB} = Re$$ \hfill (3.47)

and:

$$\frac{u_{LB}L_{LB}}{v_{LB}} = \frac{u_{phy}L_{phy}}{v_{phy}}$$ \hfill (3.48)

And that the relation between a non-dimensional system and lattice system for velocity and viscosity can be defines as:

$$u_D = \frac{\delta x}{\delta t} u_{LB}$$ \hfill (3.49)

and viscosity:

$$v_D = \frac{1}{Re} = \frac{\delta x^2}{\delta t} v_{LB}$$ \hfill (3.50)

Now, if we consider a Re,a $u_{0,LB}$ and $L_{LB}$, lattice viscosity can be obtained from equation 3.48 and can be calculated as:

$$v_{LB} = \frac{u_{0,LB}L_{LB}}{Re}$$ \hfill (3.51)

where $u_{0,LB}$ is a parameter selected according with equation 3.46. Once viscosity is calculated, we have to define the dimensional time $t_D$, but to do it we need $\delta x^2$ and $\delta t$, they can be calculated as:

$$\delta x^2 = \frac{1}{L_{LB}^2}$$ \hfill (3.52)

and with equation 3.50 :

$$\delta t = v_{LB} \delta x^2 Re$$ \hfill (3.53)

and the dimensional time $t_D$ is calculated as:
3.9 From the Lattice Boltzmann equations to the Navier-Stokes Equations

\[ t_D = \delta t \cdot TOTAL\_TIME\_STEPS \] (3.54)

We have already identified relevant parameters to move between the non-dimensional system and lattice system. A similar process is done if we want to move from the non-dimensional system to a physical system.

3.9 From the Lattice Boltzmann equations to the Navier-Stokes equations

We have already reviewed Lattice Boltzmann theory, but it is still necessary to talk about its connection with the Navier-Stokes equations. In order to understand this relation firstly we have mentioned that the distribution function must be projected on Hermite basis truncated until \( N^{th} \) order \((N=1,2 \ldots)\) and it can be written as:

\[ f(x, \xi, t) = w(\xi) \sum_{n=0}^{N} \frac{1}{n!} a^n(x, t) H^n(\xi) \] (3.55)

where \( a \) is coefficient defined as:

\[ a^n(x, t) = \int f(x, \xi, t) H^n(\xi) \] (3.56)

Now, if we obtain the Hermite polynomials based on the equilibrium function, it is possible to obtain the corresponding coefficients as:

\[ a_0^0 = \rho \] (3.57)

\[ a_0^1 = \rho u \] (3.58)

\[ a_0^2 = \rho uu + \rho(\theta - 1)I \] (3.59)

\[ a_0^3 = \rho uuu + 3\rho(\theta - 1)uI \] (3.60)

\[ a_0^4 = \rho uuuu + 3\rho(\theta - 1)^2II + 6\rho(\theta - 1)uuI \] (3.61)

Where \( \theta = 1 \) represents an isothermal fluids. These coefficients at different velocity moments up to 4th order and Boltzmann-BGK equation is used to obtain the Navier-Stokes equations through:

\[ \partial_t(a^{(n)}) + \nabla_x \cdot (a^{(n+1)} + \tilde{\rho} \nabla_x \cdot (a^{(n-1)}I - \tilde{\eta} g \cdot (a^{(n-1)}I) = -\frac{1}{\tau}(a^n - a_0^n) \] (3.62)
In order to obtain the Navier-Stokes equation it is necessary to substitute \( n=0 \) in equation 3.62 to obtain mass conservation, \( n=1 \) for Navier-Stokes and \( n=2 \) to energy equation.

### 3.10 Lattice Boltzmann solvers vs Navier-Stokes solvers

Until this point, we have defined the objectives of this work, and also we have described the conventional CFD based on the Navier-Stokes equations and the processes used for Lattice Boltzmann Method, figure 3.15, presents a summary of the main differences. However, we have not mentioned the advantages of selecting the lattice Boltzmann Method for the specific phenomena to study.

![Figure 3.15: Navier-Stokes vs LBM.](image)

In the case of the problem that we address there are some obvious advantages of LBM approach over conventional N-S CFD:

- Due to the collision is local; it is suitable for parallel computing.

- It is possible to deal with complex geometries, that includes solid moving and deformation bodies.

- The inter-phase flow interaction is more efficient.

and disadvantages:

- Computationally expensive

- Turbulence modelling

---

60
• Inherently unsteady (transient) simulations only

Despite Lattice Boltzmann method disadvantages, this methodology is suitable for the problem that we want to address due to we are going to work with two bodies, one solid and one flexible; here we can use two methodologies to represent them, staircase approximation and Immersed boundary method. In this of the fluid, we decided LBM over N-S due to we are going to work with small Re and the implementations of LBM is easier.
3.11 Summary chapter 3

In this chapter we can conclude:

- There are several possible approaches to perform computational fluid dynamics. There are solvers based on the Navier-Stokes equations and Lattice Boltzmann Method. Lattice Boltzmann method grew considerably last years due to it is highly parallelizable and it suitable to treat complex geometries.

- We have identified two alternatives to treat complex geometries on lattice Boltzmann methodology, staircase approximation and Non-Conform meshes. The staircase approach is suitable for the simple object while Non-conform meshes can handle with deformation.

- We have explained the relation among Physical units, Non-dimensional units and Lattice units.
4.1 Introduction

This chapter presents a hardware performance study, here we consider a set of experiments to identify the proper number of cores for the fastest solution as well as the adequate amount of refinement regions for accurate results.

4.2 Hardware performance study

In this section, we will describe a set of experiments designed to identify the proper balance between the number of refinement levels, number of cores and hardware selected to obtain the best solution as well as the minimum time to get the most accurate answer.

4.2.1 Hardware selection

The University of Manchester counts with a high performance computing cluster Computational Share Facilities (CSF) used by academics, post-doctoral assistants a post-graduates and managed by IT services of the University of Manchester.

CSF has 9,844 cores:

- 7,200 Intel cores with between 4GB and 5.3GB of memory per core.
- 96 Intel cores with 8GB of memory per core.
- 64 Intel cores with 16GB of memory per core.
- 48 Intel cores with 21GB of memory per core.
- 12 Intel cores with 42GB of memory per core.
CHAPTER 4. MESSAGE PASSING INTERFACE (MPI)-LATTICE BOLTZMANN METHOD AND MESH REFINEMENT STUDY

<table>
<thead>
<tr>
<th>Cases</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
<th>Case 5</th>
<th>Case 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Level 5</td>
<td>128</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Level 4</td>
<td>64</td>
<td>128</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Level 3</td>
<td>32</td>
<td>64</td>
<td>128</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Level 2</td>
<td>16</td>
<td>32</td>
<td>64</td>
<td>128</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Level 1</td>
<td>8</td>
<td>16</td>
<td>32</td>
<td>64</td>
<td>128</td>
<td></td>
</tr>
<tr>
<td>Level 0</td>
<td>4</td>
<td>8</td>
<td>16</td>
<td>32</td>
<td>64</td>
<td>128</td>
</tr>
</tbody>
</table>

Table 4.1: Characteristic length of lattice units for each level.

- 2,400 AMD cores with 2GB of memory per core.
- 24 Intel cores in GPU host nodes.
- Many GPUs.

In the case of jobs running on Intel hardware is possible to use Westmere, Sandybridge, Ivybridge, Haswell or Broadwell CPUs, and it is possible to select a specific architecture:

- Westmere provides 12 cores per node
- Sandybridge provides 12 cores per node
- Ivybridge provides 16 cores per node
- Haswell provides 24 cores per node
- Broadwell provides 24 cores per node

In this work, we use Sandybridge CPU’s for the two-dimensional flat plate where we can use less than 12 cores per node. In the case of the three-dimensional flat plate, we select we use Broadwell CPU’s where we use one or two nodes.

4.3 2D Flat plate experiments

The configuration selected for this experiment considers a computational domain of twenty times the characteristic length (flat plate length) for $X$ and $Y$ directions, $Re=300$ and a flat plate with a thickness of $th=0.0036$. The flat plate is located at the centre of a computational domain as well as refinement regions.

For these experiments, we will consider a reference of $L=128$ lattice units for the flat plate in the finest region. We will set up five different cases. Case one considers an $L=4$ in coarse but in finest $L=128$. Case two considers $L=8$ in the coarse region and $L=128$ for the finest region, but we reduce the number of levels. A similar process is followed for next cases, where it is cut the number of levels, it increases the length of $L$ in the coarse region, and $L$ remains constant for the finest region. Table 4.1 presents a summary of selected experiments.
4.3. 2D FLAT PLATE EXPERIMENTS

<table>
<thead>
<tr>
<th>Resolution</th>
<th>Levels</th>
<th>Cores</th>
<th>WC Time [hrs]</th>
<th>t dimensionless/WC Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>5</td>
<td>2x2</td>
<td>5:02:03</td>
<td>0.011035</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>2x2</td>
<td>4:10:33</td>
<td>0.013304</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>2x2</td>
<td>4:29:40</td>
<td>0.012500</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>2x2</td>
<td>5:48:19</td>
<td>0.009569</td>
</tr>
<tr>
<td>64</td>
<td>1</td>
<td>2x2</td>
<td>10:20:30</td>
<td>0.005372</td>
</tr>
<tr>
<td>128</td>
<td>0</td>
<td>2x2</td>
<td>16:05:07</td>
<td>0.003453</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>3x3</td>
<td>4:02:00</td>
<td>0.013774</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>3x3</td>
<td>2:54:49</td>
<td>0.019067</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>3x3</td>
<td>2:57:34</td>
<td>0.018762</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>3x3</td>
<td>4:49:24</td>
<td>0.011518</td>
</tr>
<tr>
<td>64</td>
<td>1</td>
<td>3x3</td>
<td>9:15:12</td>
<td>0.006003</td>
</tr>
<tr>
<td>128</td>
<td>0</td>
<td>3x3</td>
<td>15:29:50</td>
<td>0.003584</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>4x3</td>
<td>4:06:33</td>
<td>0.013519</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>4x3</td>
<td>3:24:55</td>
<td>0.016266</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>4x3</td>
<td>3:45:44</td>
<td>0.014766</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>4x3</td>
<td>5:23:30</td>
<td>0.010303</td>
</tr>
<tr>
<td>64</td>
<td>1</td>
<td>4x3</td>
<td>9:30:30</td>
<td>0.005842</td>
</tr>
<tr>
<td>128</td>
<td>0</td>
<td>4x3</td>
<td>14:00:00</td>
<td>0.003964</td>
</tr>
</tbody>
</table>

Table 4.2: Hardware test experiments 2D flat plate.

For all cases with refinement, the first level is six times L for X times and four times L for Y. The separation distance among each refinement region is one lattice cell among regions to locate more cells in finest regions. Each case is run for two hundred dimensionless time steps; we used the same hardware (Intel nodes).

It is important to comment that all these experiment for three different for three different MPI decomposition: 2x2, 3x3 and 4x3.

Table 4.2 presents a summary of results for each selected MPI decomposition, we obtain the relation between dimensionless time and a wall time clock. It is possible to see that cases without refinement process are the slowest, no matter which MPI decomposition is used.

![Figure 4.1: Hardware Study 2D flat plate](image)

In the case L=16 (three levels of refinement) and an MPI domain decomposition 3x3 presents a similar performance as L=8 (four levels of refinement) and 3x3, however for our experiments,
we have selected the four levels of refinement due to results are more accurate.

Figure 4.2 presents scheme of selected configuration for flat plate simulation, this results are shown in section 5.2.

<table>
<thead>
<tr>
<th>Cases</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Level 3</td>
<td>32</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Level 2</td>
<td>16</td>
<td>32</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Level 1</td>
<td>8</td>
<td>16</td>
<td>32</td>
<td></td>
</tr>
<tr>
<td>Level 0</td>
<td>4</td>
<td>8</td>
<td>16</td>
<td>32</td>
</tr>
</tbody>
</table>

Table 4.3: Characteristic length of lattice units for levels.

Figure 4.2: Refinement regions used for experiments for four levels of refinement.

4.4 3D Flat plate experiments

Similar to the two-dimensional case, we perform a set of experiments to obtain the fastest and accurate solution. In the 3D case, we used a three-dimensional domain of 20x20x20 times the characteristic length $L$. We selected $L=4,8,16$ and 32 lattice units for the coarse region and $L=128$ lattice units at the finest level. Table 4.9 presents a summary of selected experiments.

Due to the hardware available at the University of Manchester (Intel nodes), it is possible to use all the node with 48 cores or less. The distribution will consider three different MPI decompositions. The first configuration takes a distribution of 3x3x2, the second one 4x3x2 and the third one 4x3x4.

Like to the two-dimensional experiments, we collected the wall clock time for 20 dimensionless times. Figure 4.3 shows that MPI configuration 4x3x4 cores and $L=8$ or 16 lattice units at the coarse region are faster than other experiments.
4.5 Summary chapter 4

In this chapter we have identified:

- We have identified the hardware available at the University of Manchester. We selected the Intel Sandybridge with 12 cores per node and Intel Broadwell with 24 cores per node.

- It has presented a set of experiments to identify the fastest and accurate solution. In the case of two-dimensional flat plate immersed in a uniform flow the MPI decomposition 3x3 presents and L=8 at the coarse region, we obtain the best performance and the most accurate.

- In the three-dimensional case, the fastest MPI decomposition was 4x3x4, and L=8 is the fastest and accurate solution.

<table>
<thead>
<tr>
<th>Resolution</th>
<th>Levels</th>
<th>Cores</th>
<th>WC Time [hrs]</th>
<th>t dimensionless/WC Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>5</td>
<td>3x3x2</td>
<td>260:20:08</td>
<td>2.293x10^-5</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>3x3x2</td>
<td>110:20:12</td>
<td>5.014x10^-5</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>3x3x2</td>
<td>125:33:42</td>
<td>4.0180x10^-5</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>3x3x2</td>
<td>370:03:43</td>
<td>1.9144x10^-5</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>4x3x2</td>
<td>222:28:45</td>
<td>2.497x10^-5</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>4x3x2</td>
<td>116:26:40</td>
<td>4.744x10^-5</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>4x3x2</td>
<td>114:40:33</td>
<td>4.871x10^-5</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>4x3x2</td>
<td>207:11:33</td>
<td>2.681x10^-5</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
<td>4x3x4</td>
<td>249:50:08</td>
<td>2.223x10^-5</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
<td>4x3x4</td>
<td>76:38:24</td>
<td>7.249x10^-5</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>4x3x4</td>
<td>87:33:22</td>
<td>6.345x10^-5</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>4x3x4</td>
<td>172:05:25</td>
<td>3.228x10^-5</td>
</tr>
</tbody>
</table>

Table 4.4: Hardware test experiments on a 3D flat plate.
In this chapter, we are going to compare results obtained with LUMA for a set of selected experiments to get proper results for drag, lift and pressure coefficient for a bluff body immersed in a uniform flow. We will use the staircase approach to define the bluff body. Firstly, we are going to consider a circular cylinder immersed in a uniform flow for different Reynolds numbers from Re=2 to Re=160. Secondly, we consider a flat plate immersed in crossflow, with considering angles between $\alpha=0-60$ degrees at Re=300.

5.1 Cylinder immersed in a cross flow

An object immersed in a uniform flow has been widely studied, as this phenomenon present in many engineering applications. In most of them, the tendency is to streamline the shape to increase lift and reduce drag. One of the first attempts to understand this phenomenon is through the testing of a circular cylinder, due to their symmetry. It is known that at specific Reynolds numbers a periodic vortices shedding is developed as a consequence of the boundary layer. This regular pattern of vortices in the wake is known as Von Karman vortex street, and it correlates with Reynolds number.

The phenomenon of a cylinder immersed in a uniform flow is one of the most common experiments found in the literature. There are several possible configurations: high Reynolds, low Reynolds, free flow, confined flow, an array of cylinders or, as we see in this section, coating surfaces. There are also several possible parameters to measure, such as drag coefficient, lift coefficient, pressure coefficient or bubble size. The periodic nature of the vortex shedding phenomenon can sometimes lead to unwanted structural vibrations, especially when the shedding frequency matches one of the resonant frequencies of the structure.

In the next sections, the experiments will calculate pressure coefficient $C_p$ drag coefficient, $C_l$ lift coefficient as well as Strouhal number, $C_d$ and $St$. We consider two-dimensional cylinder
immersed in a uniform flow.

### 5.1.1 Pressure coefficient validation $C_p$

The experiments to validate LUMA results for pressure coefficient around a cylinder is based on work developed by Park in 1998 \[75\]. He presented the pressure coefficient for $Re=2-160$ and compared the results with studies published by Dennis and Chang \[29\] for $Re=5-40$ and Norberg \[72\]for $Re=130$. Where Reynolds number can was defined as:

$$Re = \frac{\rho u L}{\mu} \quad (5.1)$$

where $\rho$ is fluid density, $u$ is the free stream velocity and $L$ the characteristic length. Pressure coefficient can be calculated as:

$$C_p = \frac{2(P - P_\infty)}{\rho u_\infty^2} \quad (5.2)$$

Where $P$ is the pressure over the surface of the cylinder, $P_\infty$ and $u_\infty$ are the free stream pressure and velocity; $\rho$ is the far field density.

In this experiment, the characteristic length is the diameter $L$ of the cylinder. The considered computational domain for these sets of experiments is 50 times $L$ for $X$ and 50 times $L$ along $Y$ direction. The cylinder is located halfway in $X$ direction and slightly moved from the centre in $Y$ direction to promote the Von Karman street at higher Reynolds numbers considered. Configurations of the simulation are presented on table 5.1. We used four levels of refinement appendix A.4 presents refinement implementation process. The four levels of refinement are shown in figure 5.1, where the coarse region is marked with number 0 is the finest one with number 4. Figure 5.1 also shows an example of how looks refinement at each level.

![Figure 5.1: Four levels of refinement.](image)

Simulations will be run 150 dimensionless time steps, by this number of dimensionless time steps the integrated quantities as drag force or lift have been fully developed.

These experiments consider Reynolds $Re=2-160$. Due to we are using a simple shape like a cylinder, the immersed body is defined with the staircase approach, section 3.7.2. The advantage of this type of boundary condition (bounce-back) is that due to we are using uniform regular
5.1. CYLINDER IMMERSED IN A CROSS FLOW

<table>
<thead>
<tr>
<th>Re</th>
<th>X</th>
<th>Y</th>
<th>L_{LB}</th>
<th>U_{LB}</th>
<th>v_{LB}</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.4000</td>
</tr>
<tr>
<td>4</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.2000</td>
</tr>
<tr>
<td>10</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0800</td>
</tr>
<tr>
<td>20</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0400</td>
</tr>
<tr>
<td>40</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0200</td>
</tr>
<tr>
<td>60</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0133</td>
</tr>
<tr>
<td>80</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0100</td>
</tr>
<tr>
<td>100</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0080</td>
</tr>
<tr>
<td>120</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0067</td>
</tr>
<tr>
<td>160</td>
<td>50</td>
<td>50</td>
<td>8</td>
<td>0.1</td>
<td>0.0050</td>
</tr>
</tbody>
</table>

Table 5.1: Set up in lattice units for circular cylinder experiment.

Cartesian lattices in space, it is possible to apply them directly and it does not require another step.

All boundaries of the computational domain are assigned a far-field velocity boundary condition, which is deemed to be sufficient given the large domain size. In practise this involves the application of a uniform velocity, \( u_\infty \) at all four boundaries.

\[
 f_i = (x_b, t + \Delta t) = f_i'(x_b, t) - 2w_i \rho_w \frac{c_i \cdot u_w}{c_s^2} (5.3)
\]

where the subscript \( w \) indicates the properties located at the wall \( x_w = x_b + \frac{1}{2} c_i \Delta t \)

Figure 5.2: Cylinder configuration.

We tested ten different Reynolds numbers, firstly \( Re=2,4,10,20,40 \) and secondly Reynolds \( Re=60,80,100,120,160 \).

Figure 5.3 presents the time average pressure coefficient for the first 5 Reynolds: \( Re=2,4,10,20,40 \). These results are in good agreement with Park [75], who previously had agreed with Dennis [29].

In the case of \( Re=2 \) we observe a small difference, this performance is explained due to pressure over the surface is not computed correctly caused by staircase approach and low
Reynolds which produces a recirculation zone over this points. However, it is possible to observe that for higher Reynolds, which are the Reynolds selected for our experiments, we obtained accurate results.

Figure 5.4 presents pressure coefficient for $Re=60,80,100,120,160$. We can observe agreement with work presented by Park [75] for different $Re$.

### 5.1.1.1 Pressure coefficient at base and stagnation points

Similarly, we also measure the pressure coefficient at stagnation and base points as functions of Reynolds number, where stagnation is at $\alpha=0$ and $\alpha=180$ and corresponds with the base point.

Figure 5.5 presents stagnation of base point measurements. The pressure coefficient at stagnation point decreases as the Reynolds numbers increases as soon as we start to observe vortex shedding, (Re=50) (red line) [43]. In the case of the base point, we see a non-monotonic performance, where we see higher values of pressure coefficient $Cp$ for small Reynolds Re and low values of pressure coefficient $Cp$ for higher Reynolds Re.

The results presented in figure 5.5 were obtained with LUMA and they are marked with $\ast$ but we also show the results marked by $+$ obtained by Dennis [29], Fonberg [40], $\times$ Williamson and Roshko [92], $\wedge$ Norberg [72] and $\vee$ Henderson [46], obtaining excellent agreement with all these studies.
5.1. CYLINDER IMMERSED IN A CROSS FLOW

Figure 5.4: Time average Wall pressure coefficient. Re=60, 80, 100, 120, 160.

Figure 5.5: Base and stagnation point at different Reynolds.
CHAPTER 5. VALIDATION TEST CASE: BLUFF BODY

Figure 5.6: Time average pressure, Re=4,20,60 and 160

According to figure 5.5 at higher Reynolds number the pressure coefficient on the stagnation and base point decreases as soon as Von Karman street vortex appears. Figure 5.6 presents the time average pressure for four different Reynolds numbers: Re=4,20,60 and 160.

At all Reynolds numbers considered, we observe a high pressure at stagnation point; this high pressure indicates that the incoming flow impacts the leading edge. As soon as we increase Reynold, a vortices shedding appears, and we observe a low-pressure area at the trailing edge.

5.1.2 Drag coefficient and Strouhal number

As part of the comparative, we have measured the drag coefficient $C_d$ and Strouhal number $St$ for the same range of Reynolds numbers. Drag coefficient can be calculated as:

$$C_d = \frac{2F_x}{\rho u_\infty^2 L}$$  \hfill (5.4)

Where $F_x$ is the component in $x$ direction over immersed body, $\rho$ and $u_\infty$ are the far field density and velocity. $L$ is the characteristic length, in this case the diameter of the cylinder. On the other hand, the Strouhal number $St$ can be calculated as:

$$St = \frac{f_s L}{u_\infty}$$  \hfill (5.5)

Where $f_s$ is the frequency shedding, $L$ is the characteristic length and $u_\infty$ is the far field velocity. In the case of the St number, the Williamson [92] model ($St=-3.3265/Re+0.1816+1.6x10^4 Re$) was again used to compare it with the LUMA solution.

74
5.1. CYLINDER IMMERSED IN A CROSS FLOW

Figure 5.7: Average Cd vs Re and St vs Re.

Figure 5.7 presents the comparative for drag coefficient Cd and Strouhal number St as a function of Reynolds number Re. In the case of the drag coefficient, we can observe the same performance as Park [75] reported, where Cd decreases as Re increases and reaches an almost constant value for Re higher than 100 to 160. The corresponding figure also makes a comparison with Williamson [92], Norberg [72] and Park [75]. Similar performance as the reference papers is observed.

5.1.3 Bubble size behind cylinder

Another parameter that we measured is the time-average separation bubble length. This is the distance from the base point of the cylinder to the point where the time-average streamwise velocity is zero. Figure 5.8 presents a scheme of the relation between characteristic length L and bubble size $L_b$.

Figure 5.8: Bubble scheme.

Figure 5.9 shows a comparison with the work of Park [75], Dennis [29], Fonberg [40] and Nishioka. Here, we can observe good agreement with reference documents before and after the onset of vortex shedding.

Figure 5.10 confirms that the calculation length of the time average separation bubble is correct. We selected four different Reynolds numbers. Comparing the time average velocity for Reynolds Re=4 and 20 (before separation) with the figure 5.9, we can observe that the length of
the bubble begins to increase. For Reynolds $Re=50$ and $160$ (after separation) the time average separation bubble decreases.

### 5.1.4 Lift coefficient and Vorticity

As well as drag coefficient, lift coefficient presented a oscillatory pattern produced by vortex shedding generation. Lift coefficient can be calculated as:
5.1. CYLINDER IMMERSED IN A CROSS FLOW

\[ Cl = \frac{2F_y}{\rho u_\infty^2 L} \]  

(5.6)

where \( F_y \) is the force applied over a circular cylinder in \( Y \) direction, \( L \) the characteristic length, in this case, the diameter of the cylinder, \( \rho \) and \( u_\infty \) are the far field density and velocity.

Figure 5.11 shows the instantaneous lift coefficient over time. The oscillation of lift coefficient depends on vortex shedding; we observe that at higher Reynolds the amplitude and frequency increase in comparative with lower Reynolds. Figure 5.12 shows instantaneous vorticity. Von Karman street vortex is present at Reynolds \( Re=60 \) and 160.

Up until this point, we have compared results obtained with LUMA for a circular cylinder. In the next section, we are going to show drag and pressure coefficient for a flat plate at \( Re=300 \) and angles \( \alpha = 0 - 60 \) degrees.
5.2 Flat plate immersed in a uniform flow

As well as the case of the circular cylinder, many researchers have focused their attention on understanding the behaviour of a flat plate immersed in a uniform flow. The flat plate has been widely used to study boundary layer behaviour, and in literature, we can find a lot of experimental and numerical works.

In this section, we are going to validate LUMA code for a flat plate immersed in a uniform flow based on a paper published by Taira [84]. We will measure the mean drag coefficient $C_d$ and mean lift coefficient $C_l$ for angles $\alpha = 0-60$ degrees for a $Re=300$.

The main idea of reproducing the set of experiments proposed by Taira [84] is to verify that LUMA code can obtain accurate results ($C_p$, $C_d$ and $C_l$) for an immersed flat plate defined by staircase approach.

5.2.1 Pressure Coefficient over a flat plate

For this set of experiments, we will consider angles between zero and sixty degrees and a Reynolds number $Re=300$. The length of the flat plate is the characteristic length $L$, and the size of the computational domain is 20 times the characteristic length $L$ along $X$ and $Y$ directions. The flat plate has a thickness of $th=0.0036$, and it is located at the centre of the computational domain, where it is going to rotate the incidence angle from $\alpha = 0-60$ degrees. The immersed body uses the bounce-back boundary condition, known as a staircase approach. All boundaries of the computational domain are assigned a far-field velocity boundary condition, in other words, $U_\infty$ at

Figure 5.12: Vorticity, $Re=4,20,60$ and $160$
5.2. FLAT PLATE IMMERSED IN A UNIFORM FLOW

<table>
<thead>
<tr>
<th>Re</th>
<th>X</th>
<th>Y</th>
<th>α</th>
<th>L_{LB}</th>
<th>U_{LB}</th>
<th>v_{LB}</th>
</tr>
</thead>
<tbody>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>0</td>
<td>8</td>
<td>0.05</td>
<td>0.4000</td>
</tr>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>10</td>
<td>8</td>
<td>0.05</td>
<td>0.2000</td>
</tr>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>8</td>
<td>0.05</td>
<td>0.0800</td>
</tr>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>30</td>
<td>8</td>
<td>0.05</td>
<td>0.0400</td>
</tr>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>40</td>
<td>8</td>
<td>0.05</td>
<td>0.0200</td>
</tr>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>50</td>
<td>8</td>
<td>0.05</td>
<td>0.0133</td>
</tr>
<tr>
<td>300</td>
<td>20</td>
<td>20</td>
<td>60</td>
<td>8</td>
<td>0.05</td>
<td>0.0100</td>
</tr>
</tbody>
</table>

Table 5.2: Set up of experiments in lattice units for flat plate.

all four boundaries. Figure 5.13 presents a scheme of the configuration. Similar to the circular cylinder case, it used four levels of refinement.

Figure 5.13: Configuration of flat plate immersed in a cross flow

Table 5.2 contains a summary of the proposed experiments for a flat plate.

In figure 5.14, we can observe pressure coefficient over the length of the flat plate for different incidence angles $\alpha=10$-60 degrees, rear and front. In all cases, the rear part presents a low-pressure coefficient $C_p$, which is higher along the flat plate as soon as we increase the angle $\alpha$. In the same way, with the increase of the angle, we observe that the pressure coefficient along the flat plate grows with the incidence angle.

This behaviour was confirmed when we analysed the average time pressure. Figure 5.15 shows four different incidence angles, $\alpha=20,30,40$ and 60. In all cases, we observed high pressure at the front face while the rear face presented a low-pressure area, where the area was larger for high angles.

We also calculated the time average velocity; here we could see high velocities at the leading and trailing edges. Also, we could observe that the low-velocity area increased as we increased the incidence angle.
CHAPTER 5. VALIDATION TEST CASE: BLUFF BODY

Figure 5.14: Pressure coefficient over flat plate, $\alpha=0$-20 degrees.

Figure 5.15: Time average pressure, $Re=300$, $\alpha=20,30,40$ and 60
5.2. FLAT PLATE IMMERSED IN A UNIFORM FLOW

5.2.2 Drag and Lift coefficient over a flat plate

In this section, we present the results for the average drag and lift coefficient. In the figure 5.17 we show a comparison for seven different angles $\alpha=0$-60 and Reynolds Re=300 between Taira and Colonious [84] and LUMA results. In all cases, we obtained a good agreement with reference values.

Figure 5.16: Time Average velocity, Re=300, $\alpha=20,30,40$ and 60

Figure 5.17: Drag and Lift coefficient for $\alpha=0$–60 degrees.
In the same way, we measured instantaneous drag and lift coefficient over time, and we compare Taira and Colonious results 5.18.

Figure 5.18: \( C_d \) and \( C_l \) over time of Taira and Colonious for five different angles and \( Re=300 \).

Figure 5.19 and figure 5.20 present our results for instantaneous drag and lift coefficient. The behaviour and oscillatory shape are the same as the reference document. At low angles, we did not register oscillation while for higher incidence angles we see a periodic oscillation.

We can explain the oscillatory pattern with the generation of vortices shedding downstream flow. Figure 5.21 confirms the size of these vortices; we select four incidence angles, where for higher angles we see less frequent and larger vortices than at low angles.
5.2. FLAT PLATE IMMERSED IN A UNIFORM FLOW

Figure 5.19: Cd vs Time, $\alpha = 0 - 60$

Figure 5.20: Cl vs Time, $\alpha = 0 - 60$.
5.3 Summary of chapter 5

These findings were identified in this chapter:

- We have used LUMA code to validate a circular cylinder immersed in a uniform flow at different Reynolds numbers, Re=2-160.

- In the case of the cylinder, pressure coefficient, drag coefficient and Strouhal number, these were in good agreement with previous studies.

- Pressure coefficient at base and stagnation points were compared. Results demonstrated that as soon as vortex shedding begins, the pressure at the stagnation point begin to decrease.

- We measured the length of the time average separation bubble, obtaining accurate results.

- As mentioned in the literature, we observed vortices shedding around Reynolds Re=50.

- We modified the shape of the bluff body for a flat plate. We modified the incidence angle, but we kept a constant Reynolds Re=300.

- We compared averaged drag and lift coefficient against the literature, and we obtained appropriate results.

- Instantaneous drag and lift coefficient were monitored, and we obtained the same pattern as found in the literature.
• Pressure coefficient was measured at the front and rear sections of the flat plate. As soon as the incidence angle increased, pressure on the front increased while pressure on the back decreased.

• The size of the vortex increased at higher incidence angles. The vortex frequency decreased for higher incidence angles.

• Staircase approach and momentum exchange method have shown the proper result to calculate parameters as $C_d$ and $C_l$ for a bluff body with different shapes.
This chapter will present a set of numerical experiments of a flexible flag immersed in uniform flow, and it will be divided into two sections. The first section is an instant analysis; we are going to measure tail displacement, tail velocity as well as a power spectrum analysis. The second section considers a bifurcation behaviour analysis. The bifurcation analysis identifies the boundaries among three different states a straight state where flexible flag remains parallel to the flow, a regular flapping state where flag displace in a fixed trajectory and irregular flapping state where the flexible flag moves in an irregular trajectory.

The flexible body is defined through immersed boundary method.

### 6.1 Euler-Bernoulli beam theory

Up to this point, we have described the relationship between the Lattice Boltzmann Method and the Immersed boundary method. However, we have not clarified the theory of imposed flag deflection. LUMA uses the Euler-Bernoulli beam theory, which is a simplification of the linear theory of elasticity. This theory is used to calculate a beam’s deflection produced by loads, and it can be written as:

$$\frac{d^2}{dx^2}(EI \frac{d^2 \Delta}{dx^2}) = w$$

Where \(w\) is the force per unit or simply the distributed loading which is applied in the same direction as \(Y\) and the deflection of the beam \(\Delta x\) is the position \(X\). \(E\) is the Young’s modulus, and \(I\) is the second moment of inertia, which is calculated with the centroid of the perpendicular cross-section to the applied load. If the product of \(EI\) does not change along the beam, the equation 6.1 is simplified as:
\[ EI \frac{d^4 \Delta}{dx^4} = w \] (6.2)

Assuming that:

- The cross-section of the beam is smaller than the length.
- Any torsion or twist is neglected due to loads traversing to a longitudinal axis and passing through the centre.
- The weight of the beam can be ignored, but it must be taken into account in practice.
- The beam is considered homogeneous and isotropic; consequently Young’s modulus is constant.
- Compared with the length of the beam, deflections are assumed to be very small.
- During the bending, the cross-section remains planar and perpendicular to the longitudinal axis.
- Strain is one-dimensional, and the response is in the direction of the bending.
- Initially the beam remains straight, and deflection follows a larger circular arc when compared with the cross-section.

The bending momentum in the beam is calculated by:

\[ M = EI \frac{d^2 \Delta}{dx^2} \] (6.3)

### 6.2 Flexible flag immersed in a cross flow

The numerical simulation of a two-dimensional flexible flag immersed in a uniform flow will be compared with Jae and Huang’s experiments [52]. In this experiment, Jae and Huang were able to demonstrate that their model reproduces previous states reported in literature. As mentioned in Connell’s [27] work, it is possible for it to be characterized by three main non-dimensional parameters, the Reynolds number \( Re = uL/\nu \), non-dimensional mass ratio \( \mu = \rho_s h/\rho_f L \) and non-dimensional bending rigidity \( K_b = EI / \rho_s u^2 L^3 \).

Where for \( Re \), \( u \) is the upstream velocity, \( L \) is the characteristic length (flag’s length) and \( \nu \) kinematic viscosity, this represents the inertial forces over viscous forces. In the case of non-dimensional mass ratio (mass-per-length) \( \rho_s \) is flag density, \( \rho_f \) is fluid density, \( h \) is the thickness of the flag and \( L \) is the flat plate length.

In the case of non-dimensional bending rigidity is used the \( E \) is the Young’s modulus, \( I \) is the second moment of inertia, \( \rho_s \) is the flag density, \( u \) is fluid velocity and \( L \) the characteristic length.
6.2. FLEXIBLE FLAG IMMERSED IN A CROSS FLOW

(flag's length). In most of the realistic flows the bending rigidity is very low and in consequence flag's dynamics is primarily governed by mass ratio and Reynolds number.

In the next section, we are going to describe a set of numerical experiments used to validate a flag immersed in uniform flow.

6.2.1 Instantaneous analysis

The experiments presented by Lee and Huang [52] consisted in a rectangular domain, the uniform flow moves from left to right. At the inlet, a Dirichlet boundary condition \((u=U_\infty, v = 0)\) and far-field boundary conditions, at the outlet a convective boundary condition was used. The computational domain used was eight by eight times the length of the flag. For their instantaneous analysis, Lee and Huang defined a constant \(Re=200\), a bending rigidity \(K_B = 10^4\) and range mass ratio \(0.1 \leq \rho \leq 3.0\). The initial angle of the flag is \(\alpha=18\) degrees. The two-dimensional flag is clamped upstream on the leading edge, while the trailing edge on the downstream is left free. Figure 6.1 presents a scheme of a flexible flag immersed in a cross flow.

![Figure 6.1: Flexible filament immersed in a cross flow.](image)

Figure 6.1 shows a summary of the numerical experiments selected to compare their results with LUMA code.

![Figure 6.2: Proposed experiments for bifurcation behaviour analysis.](image)

The computational domain considered to use with LUMA simulation was: twelve times \(L\) (flag's length) in the \(X\) direction and ten times in \(Y\) direction. The flexible flag is located at the centre computational domain. Figure 6.3 presents a summary. Similar to cylinder case presented in section 5.1 we use four levels of refinement. Far-field velocity \(U_\infty\) boundary conditions are used for all boundaries. The Reynolds number used was \(Re=200\), a bending rigidity \(K_B = 10^4\) and range mass ratio \(0.1 \leq \rho \leq 3.0\) and the initial angle of the flag is \(\alpha=18\).

Similar to Lee and Huang [52] all these cases were run for 100-time steps until flow developed.
6.2.2 Tail position analysis

In this section, we will analyse and compare the vortex dynamics produced by the flexible flag. Here, we will see that the uniform flow that will move along the domain and it will create the initial flapping of the flexible flag, which occurs by a change of momentum generated by forces exerted on the flag.

The flapping of the flag is influenced by the relationship between the non-dimensional mass ratio, Reynolds number and non-dimensional bending rigidity. The size and shape of vortex depend on the trajectory and the amplitude of the filament’s displacement.

To verify that LUMA is reproducing a flexible flag correctly, we have to identify the distinct regimes of response. Connell defines these regimes as (I) fixed point stability; (II) limit-cycle flapping and (III) chaotic flapping or by Huang (I) straight–state, (II) regular flapping state and (III) irregular flapping State. In regime (I) the body remains steady straight with no displacement of the trailing edge. In regime (II) the response of the body settles to a period-one limit-cycle oscillation of constant frequency and amplitude. Regime (III) exhibits at the trailing edge a non-periodic behaviour characteristic of chaos.

Firstly, in figure 6.4, we are going to show the cases for $\mu=0.1,0.2,0.8,2.0$ and $3.0$, a $Re=200$ and bending rigidity $K_b=0.0001$. We measured instantaneous tail position (Y displacement), tail velocity $dy/dt$ and power spectra.

In the case of $\mu=0.1$, we can see that despite the initial perturbation produced by the initial angle, the tail remains parallel to the fluid, this state is known as fixed point stability, means that the flag remains static and the power spectra is not registered. It is important to mention that the objective of power spectra analysis is to identify more than one signal in Y displacement, in other words, if power spectra identify more than one signal means that the displacement of the flexible flag can be defined as the irregular flapping state.

When the mass ratio is $\mu=0.2$, the tail presents a regular displacement, its deformation in tail velocity remains constant, and we observe a single domain frequency. As Jae and Huang’s [52] paper, LUMA reproduces the regular flapping state. In this case, we can see the Von Karman street vortex behind the flag’s wake and is presented in figure 6.5. This narrow vortex shedding
Figure 6.4: Tail position analysis (from top to bottom, \( \mu = 0.1, 0.2, 0.8, 2.0, 3.0 \)), a) Y displacement, b) Tail velocity \( \frac{dy}{dt} \) and c) Normalized power spectra.
is proportional to tail displacement amplitude.

When we increase the mass ratio to $\mu=0.8$, the tail rises its displacement as well as velocity. Under these conditions, the regular flapping state is reproduced. In this case, we observe a small difference with Jae and Huang [52] because we find a second low frequency, this could indicate that our LUMA model starts the transition to an irregular flapping state. However, it is possible to observe a dominant frequency. In this case, the deformation is more pronounced, figure 6.6 and a second vortex appears when the flap of the flag changes its direction.

For $\mu=2.0$ it is clear that the transition to the irregular flapping state is due to the additional peak frequencies. Here the amplitude of tail displacement and velocity increase in comparison to smaller mass ratios. Figure 6.8 shows, in a similar way as $\mu=0.8$, that primary and secondary vortices are created as a downward sweep event, however vortices present different sizes and strengths.

When mass ratio $\mu=3.0$, the system can be considered as chaotic, this is the irregular flapping state, the tail displacement increases but it is possible to see several peak frequencies produced by the tail’s acceleration. This phenomenon is known as a snapping event, which is characterised by the rapid change in drag, lift, tension and velocity. In this case, despite having an irregular flapping state in the reference paper, the behaviour is different. While Jae and Huang [52] present the creation of three vortices, the first one was created during the earlier down-sweep of the flag, the second one when the flag swept upward and the third one when the tail of the flag gradually rolled up over time; LUMA shows that vortices remain attached to the flag, producing larger vortices moving vertically.
Figure 6.6: Time-evolving vorticity contours of the flow surrounding the flag for $\mu=0.8, \text{Re}=200$ and $K_b = 0.0001$.

Figure 6.7: Time-evolving vorticity contours of the flow surrounding the flag for $\mu=2.0, \text{Re}=200$ and $K_b = 0.0001$. 
6.2.3 Bifurcation behaviour

In the previous sections, we commented that the flapping of a flag could be characterised by three main parameters; non-dimensional mass ratio, non-dimensional bending rigidity and Reynolds number. In this sense, several works have been tried to predict flapping flag behaviour for \( \text{Re}=200 \), for example, Watanabe, Connell and Yue, Alben and Shelley, Eloy et al., and Jae and Huang. Moreover, Connell and Yue also presented a model based on a potential flow theory in a linear stability analysis and the flapping criteria can be defined as:

\[
\frac{\rho}{\pi \rho + 1} > 1.328 \text{Re}^{1/2} + 4\pi^2 U_s^{-2}
\]  

(6.4)

Where \( U_s \) is the reduced velocity and is related to non-dimensional bending rigidity and can be calculated as:

\[
U_s = \frac{1}{\sqrt{K_b}}
\]  

(6.5)

The intention of equation 6.4 is to define limits among three different dynamic states of a flexible flag: \textit{I)} straight–state, \textit{II)} regular flapping state and \textit{III)} irregular flapping state. Figure 6.9 presents bifurcation behaviour for \( \text{Re}=200 \). Figure 6.9 presents the different models that simulate the behaviour of a flexible flag, Connel, Alben, Michelin, Eloy present their limit between \textit{I)} straight–state and \textit{II)} regular flapping state, while Huang model presents a limit between \textit{II)} regular flapping state to \textit{III)} irregular flapping state.
We have performed a set of numerical experiments to identify the limits for three different states. Here, it is possible to see that at low reduced velocities, no matter which mass ratio is selected, the flexible flag remains straight. Similar to the document presented by Jae and Huang [52], the LUMA model shows that for small mass ratios, no matter which reduced velocity is selected, the flexible flag remains parallel to the flow. In the transition region straight state, the regular flapping state LUMA model overpredicts the boundaries; this can be explained as being due to Linear stability analysis not considering the effects of vortex shedding.

In the case of the limit from regular state to the irregular state, the LUMA model presents the transition at small mass ratios in comparison to that presented by Jae and Huang [52]. This behaviour confirms the difference in irregular flapping state and vortex shape presented in section 6.2.2 and figure 6.8.

We have already reproduced numerically the experiments presented by Jae and Huang [52]. LUMA model has been able to reproduce similar results as the selected model, but most importantly, we have obtained the different possible states under the same conditions.

### 6.3 Summary of chapter 6

In this chapter, we have identified that:

- We reproduced the same numerical experiment as presented in the literature by Jae and Huang [52].

- We developed a tail position analysis. We measured tail displacement, tail velocity and gave a frequency analysis.

- Immersed boundary method was able to simulate the behaviour of the flexible flag.
• LUMA implementation was able to reproduce states previously reported in the literature.

• For mass ratio $\mu=0.1$, the tail remains parallel to the flow. If mass ratio $\mu$ increases the displacement of the tail increases. For mass ratio $\mu=0.1, 0.2, 0.8, \text{ and } 2.0$ we obtained similar results as the reference work, but for mass ratio $\mu=3.0$, even though we have an irregular flapping behaviour the tail displacement is different. This could be by the fact that the mass ratio used by Jae and Huang [52] did not consider the flag’s thickness.

• When we observe an irregular flapping state, the power spectra analysis registers multiple frequencies.

• We have also shown the vortices shedding from the flexible flag. A small mass ratios the size of vortices remains small, but as soon as we increase the mass ratio the displacement of the tail gets more substantial, and in consequence, the size of the vortices increases but their frequency reduces.

• We performed a bifurcation behaviour analysis. We tested several combinations of mass ratio $\mu$ and reduced velocity, which is related to bending rigidity as found in equation 6.5.

• We identified the limit between a straight-state to a regular flapping state and from a regular flapping state to an irregular flapping state.

• Similar to that found in the reference work, for low reduced velocities $U_s$ and any mass ratio $\mu$ the flexible flag, remains parallel to the flow.

• The model implemented in LUMA overpredicts the limits in comparison with the Linear Stability Analysis presented by Connell [27].
This chapter presents a set of experiments selected to understand the performance of a flexible flag behind a flat plate. In chapters 5 and 6, we have reproduced two experiments, firstly we have demonstrated that LUMA code can obtain proper results for drag, lift and pressure coefficient for different angles in a flat plate. Also, we have reproduced the performance of a flexible flag for a constant Reynolds number, bending rigidity $K_B$ and several mass ratios $\mu$. We reproduce the experiments proposed by Lee and Huang [52] and LUMA code is able to reproduce the three different states for reported, (I) straight–state, (II) regular flapping state and (III) irregular flapping state.

Based on experiments presented in chapter 5 and 6, we are going to test a flexible flag attached to a flat plate. In the first section of this chapter, we are going to reproduce the Linear Stability Analysis (LSA) for three Reynolds numbers $Re=100,200$ and 400 to identify and avoid the points where flexible flag response with an irregular flapping state.

Once that we know the response of the flexible flag, we are going attach the flexible flag to a flat plate, and we will present a flapping frequency analysis in order o identify the points where the flexible flag has an interesting performance. With this information, we will analyse the vortex shedding produced by the movement of the flexible flag and we will present a tail displacement analysis to know the possible states. To finish, we will measure the drag and pressure coefficient.

In our proposed experiment we will consider different at Reynolds numbers $Re$, mass ratios $\mu$ and reduces velocities $U_s$.

### 7.1 Bifurcation behaviour analysis, Reynolds $Re=100,200,400$

In this section, we are going to use the same configuration as presented in section 6. The computational domain is twelve times $L$ (flag's length) in the $X$ direction and ten times in $Y$ direction
and four levels of refinement. A constant velocity was used at the inlet and far field for top bottom and outlet. The Reynolds number selected for these experiments were \( Re=100,200 \) and \( 400 \).

The main idea of this set of experiments is to identify limits among three different possible states for a constant Reynolds. We are going to change reduced velocity \( U_s \) in combination with mass ratio \( \mu \). Figure 7.1 presents a summary of the experiments proposed.

![Figure 7.1: Proposed experiments to identify limits among different flapping states.](image)

These experiments will give us an idea of where are the limits among the three different states for a constant Reynolds, in this way we can avoid the irregular flapping area and in case of having an irregular flapping state will be just by the action of vortices shed from the flexible plate.

### 7.1.1 Reynolds \( Re=100 \), Mass ratio \( U_s=20-100 \) and Reduced velocity \( \mu=0.05-10 \)

According to the set of experiments mentioned in figure 7.1, for a Reynolds \( Re=100 \), we selected reduced velocities between \( U_s=20-100 \) and mass ratios \( \mu=0.05-10 \). As a reference, we included the Linear Stability Analysis developed by Connell [27].

The Figure 7.2 shows the results for selected parameters. At a Reynolds \( Re=100 \), the limit between the straight-state and regular-state moves slightly to the right in comparison with Reynolds \( Re=200 \), figure 6.9. In this case, the safety range to avoid irregular flapping state is reduce velocity among \( U_s=20-35 \) and mass ratios among \( \mu=0.05-2 \). When mass ratio \( \mu=0.2 \), we can still be considered as part of the straight-state. We can observe that for this Reynolds, the regular-state appears around a mass ratio \( \mu=0.3 \) and high reduced velocity \( U_s=50 \). In the case of the irregular-state, we see the beginning approximately at mass ratio \( \mu=1 \) and \( U_s=75 \).

For this experiment, we did a frequencies and amplitudes analysis of the tail displacement. We built a frequency and amplitude surface. The right part of figure 7.3 shows the relation between flapping frequency, mass ratio \( \mu \) and reduced velocity \( U_s \). The right side of the figure 7.3 shows that at lower mass ratios than \( \mu=0.2 \), the flexible flag remains parallel to the flow. Around a reduced velocity \( U_s=50 \) and mass ratio \( \mu=0.3 \), we see the beginning of high flapping frequency, which is precisely the position of the limit between straight-state and regular-state. From this point, we see a fall in flapping frequency, where larger mass ratios (irregular flapping state) presents the lowest flapping frequencies. The behaviour previously described is explained
7.1. BIFURCATION BEHAVIOUR ANALYSIS, REYNOLDS RE=100,200,400

7.1.1 Reynolds Re=100, Mass ratio \( U_s = 20-100 \) and Reduced velocity \( \mu = 0.05-10 \)

Figure 7.2: Bifurcation behaviour Re=100.

By the fact that the regular flapping state presents just one natural frequency, while the irregular flapping state presents multiple frequencies.

In the case of the right side of the figure 7.3, we show the amplitude surface. Here, we can see that higher mass ratios \( \mu \) present the higher amplitudes, while low mass ratios \( \mu \) present the lower amplitudes. Only, we can say that the regular-state presents high frequencies and small amplitudes, while high amplitudes and low frequencies characterise the irregular state.

7.1.2 Reynolds Re=200, Mass ratio \( U_s = 20-100 \) and Reduced velocity \( \mu = 0.05-10 \)

Similar to the last section, we are going to consider a constant Reynolds Re=200, and selected reduced velocities between \( U_s = 20-100 \) and mass ratios \( \mu = 0.05-10 \). The Linear Stability Analysis is located at the reference. In this case, the straight-state is narrow in comparison with Reynolds Re=100. For this experiment, the safety range to avoid irregular flapping state is to reduce velocity among \( U_s = 20-100 \) and mass ratios among \( \mu = 0.05-1 \). The regular flapping state initialises
at a mass ratio $\mu=0.2$ and a reduced velocity $U_s=35$.

For a Reynolds $Re=200$, we show in figure 7.5 the relation between flapping frequency and flapping amplitude with the mass ratio $\mu$ and reduced velocity $U_s$. The left side of the figure 7.5 shows the flapping frequency, here the straight-state ends around a mass ratio $\mu=0.1$. From $\mu=0.2$ to $\mu=0.5$ we observe the highest frequencies and the regular flapping state.

The irregular flapping state begins at a mass ratio $\mu=1$ and a reduced velocity $U_s=100$. In the case of Reynolds $Re=200$, the regular flapping state presents high frequencies and low amplitude, while irregular-states show the low frequency and high amplitude.

### 7.1.3 Reynolds $Re=400$, Mass ratio $U_s=20-100$ and Reduced velocity $\mu=0.05-10$

For a Reynolds $Re=400$, we also mapped the limits between different possible states. The limit between the straight-state and the regular flapping state is now located around a mass ratio $\mu=0.1$ and reduced velocity $U_s=50$. The irregular flapping state is reached at approximately $\mu=0.1$.

![Figure 7.4: Bifurcation behaviour Re=200.](image)

For a Reynolds $Re=400$, we show in figure 7.5 the relation between flapping frequency and flapping amplitude with the mass ratio $\mu$ and reduced velocity $U_s$. The left side of the figure 7.5 shows the flapping frequency, here the straight-state ends around a mass ratio $\mu=0.1$. From $\mu=0.2$ to $\mu=0.5$ we observe the highest frequencies and the regular flapping state.

The irregular flapping state begins at a mass ratio $\mu=1$ and a reduced velocity $U_s=100$. In the case of Reynolds $Re=200$, the regular flapping state presents high frequencies and low amplitude, while irregular-states show the low frequency and high amplitude.

![Figure 7.5: (Left)Flapping frequency, (Right)Amplitude for selected mass ratios $\mu$ and reduced velocities $U_s$.](image)
7.1. BIFURCATION BEHAVIOUR ANALYSIS, REYNOLDS RE=100,200,400

and $U_s=50$. The case of Reynold Re=400 presents a larger area for an irregular flapping state than regular state or straight-state.

![Figure 7.6: Bifurcation behaviour Re=400.](image)

The frequency and amplitude surface were calculated for Re=400. As with the previous cases, the flapping frequency increased as soon as we crossed the limit and from then on started to decrease.

In this case, the area of higher flapping frequency is smaller than in previous cases. Here, as well as in most cases presented until now, no matter the Reynolds number, the straight-state does not register any flapping frequency; the regular flapping state shows the maximum frequency for mass ratios close to the limits and with low amplitude, while the irregular flapping state presents low frequency and high amplitude.

Up until this point, we have identified the limits of the model implemented in LUMA. We know what kind of performance the flexible flag will have under the selected parameters. We understand the relation between flapping frequency, flapping amplitude and the three different

![Figure 7.7: (Left) Flapping frequency, (Right) Amplitude for selected mass ratios $\mu$ and reduced velocities $U_s$.](image)
possible states. In the next section, we will present a set of experiments to understand the effect of a flat plate at different angles and different Reynolds numbers.

### 7.2 Flexible flag behind a flat plate

In chapter 2, we commented on several attempts to understand the performance of a flexible flag behind a bluff body. In most of these experiments, the bluff body is a circular cylinder, and there is a gap for different bluff body shapes. We have decided to use a flat plate due to we want to understand the influence of the incidence angle, the Reynolds numbers $Re$, mass ratios $\mu$ and reduces velocities $U_s$ over downstream flow.

The configuration selected for our experiments considers a computational domain of twenty times for $X$ and $Y$ the characteristic length, in this case, is the length of the flat plate. The flat plate is located at the centre of the computational domain, where it is attached to the same length flexible flag. The angle of the flat plate $\alpha$ will take values between 20 and 40 degrees $\alpha=20-40$. In the case of the flexible flag, the initial angle $\beta$ will be 18 degrees. We use bounce-back boundary conditions to define the flat plate, section 3.7.2, and immersed boundary method for flexible flag, section 3.7.3.1.

At the inlet, it is considered a uniform flow while the top, bottom and outlet use far field velocity boundary condition $U_\infty$. Similar to our previous cases, we considered four levels of refinement. Figure 7.8 shows a scheme of the selected configuration.

![Figure 7.8: Flat plate and flexible flag configuration.](image-url)

As mentioned before, the performance of a flexible flag modifies with three parameters: Reynolds number $Re$, mass ratio $\mu$ and reduced velocity $U_s$. For our experiments, we are going to consider
three different Reynolds Re=100,200 and 400, four different reduced velocities $U_s=5,10,30,40$ and four different mass ratios $\mu=0.05,0.1,0.2,0.3$. According to the experiments presented in the previous section, under these conditions the flexible flag response, with straight-state or regular flapping state, the irregular flapping state is discarded. Figure 7.9 shows a summary of selected experiments.

![Figure 7.9: Set of experiment proposed.](image)

In the next section, we will describe the results of select experiments.

### 7.2.1 Flapping frequency analysis

In the last section, we commented that irregular flapping state was discarded for our study, that means that under the selected parameters the flexible flag must keep parallel to the flow in absent of the flat plate. Due to we do not know how flexible flag will response to the incidence angle and Reynolds number, we performed a frequency analysis, which will help us to identify the points where displacement is registered.

In order to reduce our experiments and detect interest points, we are going to organize our tests in relation with incidence angles, this means that at each set of experiments, we will consider one incidence angle $\alpha$, three Reynolds numbers $Re$, four reduced velocities $U_s$ and four mass ratios $\mu$. It is important to mention that a tail displacement analysis as presented in chapter 6 was carried out for all presented experiments.

#### 7.2.1.1 First set of experiments: $\alpha=20$, $Re=100-400$, $U_s=5-30$ and $\mu=0.05-0.3$

The figure 7.10 shows the tail displacement frequency with the corresponding Reynolds number $Re$, reduced velocity $U_s$ and mass ratio $\mu$. In this figure, we can see for Reynolds lower than $Re=400$, flapping frequency registers a small displacement or not register any movement at all. In
other words, for these conditions, the flexible flag will remain at the same position, and incidence angle or Reynolds number will not affect its performance.

A different performance is captured when the Reynolds number \( Re = 400 \). In this case, we registered similar flapping frequencies no matter which reduced velocity \( Us \) or mass ratio \( \mu \) is used. Here it is important to remark that the incidence angle of the flat plate has modified the performance of the flag. Based on the experiments presented for Reynolds \( Re = 400 \) in figure 7.6, at mass ratios lower than \( \mu = 0.1 \), the flexible flag must remain in the straight-state; however it presents a displacement, this can be confirmed when flow properties are analysed in section 6.2.3.

### 7.2.1.2 Second set of experiments: \( \alpha = 30 \), \( Re = 100-400 \), \( Us = 5-30 \) and \( \mu = 0.05-0.3 \)

For this set of experiments, we modified the incidence angle \( \alpha \) for the flat plate, the figure 7.11 presents flapping frequency analysis. In the case, when Reynolds \( Re = 100 \), it was not observed any displacement. When Reynolds number is modified to \( Re = 200 \), a movement is detected in all selected points. In comparative with figure 7.4 the incidence angle modifies the performance of the flag due to smaller values than \( \mu = 0.2 \) remain parallel to the flow in flat plate absent.

The case of Reynolds \( Re = 400 \) registers a displacement as well. However, when reduced velocity \( Us = 20 \) or 30 and mass ratio \( \mu = 0.3 \) the frequency falls. The area where this performance is present reach the limits between straight-state and regular-state 7.4, this can suggest that at points we can have an irregular flapping and we have multiple frequencies or a new dynamical state.
7.2. FLEXIBLE FLAG BEHIND A FLAT PLATE

7.2.1.3 Third set of experiments: \( \alpha=40 \), \( \text{Re}=100-400 \), \( U_s=5-30 \) and \( \mu=0.05-0.3 \)

The case when we use an incidence angle \( \alpha=40 \) is the most complex due to the flapping frequencies registered present upward and downward trends. At reduced velocity \( U_s=5 \) for all mass ratios the frequency increase as a function of Reynolds. When the reduced velocity is \( U_s=10 \), we observe a fall in frequency for all Reynolds numbers. When reduced velocity is \( U_s=20 \), we register a frequency again.

Figure 7.12: Flapping frequency \( \alpha=40 \), \( \text{Re}=100-400 \), \( U_s=5-30 \) and \( \mu=0.05-0.3 \).

Figure 7.12 shows a summary of all the experiment performed. In the next section, we are going to analyse the flapping performance capture in some of our tests.
7.3 Tail displacement analysis

We have already identified the points where the flexible filament responds to the angle and the Reynolds number. For all these cases we have developed a tail displacement analysis due to we have identified some interesting performance.

In the previous section, we identify that when the angle $\alpha = 20$ and Reynolds number $Re=100$ or 200 any flag displacement is registered, no matter which mass ratio is used. A similar performance was detected when angle alpha $\alpha = 30$ and Reynolds number $Re=100$, however as soon as we increase the Reynolds $Re=200,400$ the flexible flag start response to the downstream flow.

In next sections, we are going to analyse a tail displacement analysis for an angle $\alpha=40$, a fixed mass ratio $\mu=0.1$, two Reynolds numbers $Re=200$ and 400, and four reduced velocities $U_s$, due to under these values an interesting performance was observed.

Before to describe the experiments, it is important to know, how is the performance of the flat plate with an angle $\alpha=40$ and Reynolds $Re=400$ without a flexible flag and with a rigid flag acting as splitter plate.

The figure 7.13 presents the instantaneous vorticity for a flat plate at Reynolds $Re=400$, where vortices roll into the low-pressure area behind it, this vortex grow until separation is done. A similar performance is presented by the superior part of the flat plate. Here, we observe a constant vortex production.

![Figure 7.13: Instantaneous vorticity flat plate $\alpha=40$, Re=400](image)

A different performance is observed as soon as we locate the rigid plate, which acts as a splitter plate. The main function of a splitter plate is to suppress vortices shedding and reduce drag. In comparative with 7.14, we see that the splitter plate modified the shape of the vortices.
7.3. TAIL DISPLACEMENT ANALYSIS

as well as its frequency. Here, we can see a larger low-pressure area, and the formation of the vortex is done far from the back of the flat plate.

Figure 7.14: Instantaneous vorticity flat plate and rigid flexible flag $\alpha=40$

In the next section, we perform a tail analysis frequency analysis, and we will explain the relation of it with instantaneous vorticity.

7.3.1 $Re=200$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$

The figure 7.15 shows the tail displacement analysis for $Re=200$, $\alpha=40$, $\mu=0.1$ and four reduced velocities, $U_s=5-30$. When reduced velocity $U_s=5$ (top), figure 7.15, we observe a regular flapping state, this is characterised by a single frequency and a constant velocity. This is confirmed with a single frequency registered during the power spectra analysis.

To identify the type of displacement for selected experiments, we are going to show instantaneous vorticity. The figure 7.16, shows the instantaneous vorticity for reduced velocity $U_s=5$. Here, we can confirm the regular displacement showed in figure 7.15. The flexible flag displaces uniformly and does not lose their shape. Under these conditions, the flexible flag performs as a moving splitter plate, reducing the vortex shedding production.

As soon as we increase the reduced velocity $U_s=10$, a small displacement is registered. This state of the flexible flag under current conditions presents a state that we had not seen previously. The flexible flag remains,7.17, a single position due to is trapped, and it does not show any deformation. We can see that the formation of vortices does not permit that flag return to the original position. Here, although vortex production is reduced, they remain attached to the bluff body and flexible flag. Under current conditions, flexible flag suppresses vortex shedding.
CHAPTER 7. PROPOSED EXPERIMENTS AND DISCUSSION

Figure 7.15: Tail displacement analysis, Re=200, $\alpha=40$, $U_s=5-30$ (top-bottom) and $\mu=0.1$

Figure 7.16: Instantaneous vorticity Re=200, $\alpha=40$, $U_s=5$ and $\mu=0.1$
When the reduced velocity takes higher values, $U_s=20$, we register a new displacement. At this point, the flexible flag can leave the zone where it was trapped. Under this condition is the first time that we see that the action of the free stream deforms flexible flag. Figure 7.15, (third set of figures from top to bottom), show Y-direction displacement with a dominant frequency. Here, we observe that vortex shedding increase compared with rigid case. In contrast with previous cases, the low-pressure area is smaller producing higher interaction among vortices.
When reduced velocity increase $U_s=30$, we observe a similar performance as $U_s=20$, the flexible flag has a displacement, and the action of the fluid deforms it. If we observe figure 7.15 the power spectra analysis presents more than one signal, this performance means that flexible flag displaces irregularly.

![Instantaneous vorticity](image)

Figure 7.19: Instantaneous vorticity Re=200, $\alpha=40$, $U_s=30$ and $\mu=0.1$

Figure 7.19 shows vortices create by the action of the flexible flag, with the displacement of the flag low-pressure area decrease and vortices interact each other before to be shed.

### 7.3.2 $\text{Re}=400$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$

We have analysed the performance of the flexible flag when the Reynolds number Re=200, it is time to present the experiments when Reynolds Re=400. The figure 7.20 shows the results for Re=400, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$.

When the reduced velocity $U_s=5$, we observe a regular displacement, constant velocity and a single frequency is registered. For these parameters, we observe that the flexible flag does not present any deformation, however in comparison with lower Reynolds the flexible flag displacement is higher. Under these conditions, the flexible flag displaces but do not suffer any deformation, figure 7.21.

When reduced velocity is $U_s=10$, figure 7.20, we observe a small displacement at the trailing edge. The flag is deflected until some point, and it does not recover its initial position, and it does not suffer any deformation, the flexible flag remains trapped by vortex shedding. However, for this Reynolds number, we can observe that vortices roll into the bifurcated area, these
7.3. TAIL DISPLACEMENT ANALYSIS

Figure 7.20: Tail displacement analysis, Re=400, $\alpha=40$, $U_s=5-30$ (top-bottom) and $\mu=0.1$

Figure 7.21: Instantaneous vorticity Re=400, $\alpha=40$, $U_s=5$ and $\mu=0.1$
CHAPTER 7. PROPOSED EXPERIMENTS AND DISCUSSION

phenomena produce a larger vortex that separates into small vortices. Figure 7.22 presents the vortex shedding.

As soon as we increase the $U_s=20$, we observe a larger displacement with a constant frequency. Here, it is possible to see the action of vortices deforms that flag. These conditions increase vortex shedding in comparative with previous cases. Figure 7.23 shows how vortices interact with flag producing its deformation.
When the reduced velocity $U_s=30$, figure 7.24, the power spectra analysis registered more than one signal, this performance suggests an irregular flapping state. Figure 7.24 presents instantaneous vorticity. Here we observe that flexible flag interacts with vortex shedding, reducing the low-pressure area and increasing vortex production.

![Figure 7.24: Instantaneous vorticity Re=400, $\alpha=40$, $U_s=30$ and $\mu=0.1$](image)

We have already analysed tail displacement, we have shown the performance of a flat plate alone and the effect of attaching a splitter plate, also we have described the performance of a flexible flag under different conditions. In the following section, we are going to quantify pressure and drag coefficient with all cases presented previously.

### 7.4 Pressure coefficient $C_p$

Previously, we mention that a splitter plate works as a mechanism of drag reduction. In this section, we will analyse the pressure coefficient for the selected experiments. Firstly, we are going to present the results for Reynolds $Re=200$ and later for $Re=400$.

#### 7.4.1 Pressure coefficient, $Re=200$, $\alpha=40$, $U_s=5-30$ and $\mu=0.1$

In this section, we are going to describe our results for the pressure coefficient, but firstly we are going to comment on the time average pressure. The top right presents the average time pressure for flat plate alone. In this case, we observe a low pressure on the rear part of the flat plate; this performance promotes that vortices occupied this space and as we see in figure 7.13.

The top left presents the average pressure for a rigid flag, in this case, we observe a lower pressure area, but it is not as low as the case without any modification. When we use flexible flags,
CHAPTER 7. PROPOSED EXPERIMENTS AND DISCUSSION

Figure 7.25: Time average pressure, Re=200, $\alpha=40$

we see the low-pressure area behind the plate as well, however for reduced velocities, $U_r=20$ and 30 the rear pressure is lower than $U_r=5$ and 10. Once presented the contours for time-averaged pressure, we can comment about the pressure coefficient in figure 7.26.

In the figure 7.26, we can see the calculation for rear and front pressure coefficient. As we can see time average pressure, there is not a significant difference for the front face; the main difference is present at the rear face. The lowest pressure is presented when the flat plate does not suffer any modification. The cases that use larger reduced velocities present lower pressures on the rear face but not as the case of plate alone. This performance confirms what we observe in time average pressure contours.

The case with the lowest performance in the case of the rigid plate, followed by the cases of reduced velocities $U_r=5$ and later $U_r=10$. 

114
7.5. DRAG COEFFICIENT.

7.4.2 Pressure coefficient, Re=400, α=40, U_s=5-30 and μ=0.1

Similarly, in this section, we are going to analyse the pressure coefficient for Re=400. Firstly we are going to observe the time average pressure contours. The figure 7.28 shows the pressure contours for Re=400, where the top right shows the flat plate without any modification. This case presents the lowest pressure on the rear face and the case where the flag works as the splitter plate presents the highest pressure on the rear face.

In the figure 7.27, we can see that the highest reduced velocity the lower time average pressure on the rear face. Now, if we analyse pressure coefficient in figure 7.28, we can see that similar to the case when the Reynolds Re=200, the highest low pressure appears in the flat plate without modification, the lowest pressure coefficient is for the case of the rigid plate acting as splitter plate. In the cases where we modified the reduced velocity, we see similar distribution as Re=200, where the highest reduced velocity the lowest pressure coefficient.

We have already reviewed the pressure coefficient; now we have to evaluate the drag coefficient.

7.5 Drag coefficient,

In this section, we are going to present the instantaneous drag coefficient and average drag coefficient, and what is happening, we will increase the Reynolds number. We will analyse the how is the instantaneous drag coefficient is modified and what is happening with the average value.
Figure 7.27: Time average pressure, \( Re=400, \alpha=40 \)

Figure 7.28: Pressure coefficient \( Re=400, \alpha=40, U_s=5-30, \mu=0.1 \).
7.5. DRAG COEFFICIENT

7.5.1 Drag coefficient, \( Re=200-400, \alpha=40, U_s=5-30 \) and \( \mu=0.1 \)

The figure 7.29 presents the instantaneous vorticity for all experiments. In this figure, we can see how the instantaneous drag coefficient is modified by the action of the flexible and rigid flag. The case, where the flag remains rigid, presents smaller oscillation in comparative with cases where the flag is flexible.

![Figure 7.29: Instantaneous drag coefficient \( Re=200 \) and \( \alpha=40, U_s=5-30, \mu=0.1 \)](image)

Similar performance can be observed when Reynold \( Re=400 \). In this case, the instantaneous drag increased in all cases. However, the rigid flag still presents the lower oscillation in comparative with the other cases.

![Figure 7.30: Instantaneous drag coefficient \( Re=400 \) and \( \alpha=40, U_s=5-30, \mu=0.1 \)](image)

We have calculated the averaged drag coefficient over the flat plate for all our experiments.
Figure 7.31 compares the all cases. When the flat plate does not suffer any modification it is registered a maximum averaged drag coefficient, as soon as we use the rigid splitter plate, we observe an important reduction in averaged drag coefficient.

![Figure 7.31: Averaged drag coefficient vs Reduced velocity.](image)

When we introduce the flexible flag, \( U_s = 5 \), we observe an increase over the average drag coefficient. It is important to comment that under these conditions the flexible flag does not suffer any deformation and it acts as a moving splitter plate, and its displacement can be categorised as a regular flapping state.

When reduced velocity increase \( U_s = 10 \), the flexible flag remains trapped by vortices, here we observe a suppression in vortex generation, and despite the fact that we measure a reduction in average drag coefficient the rigid splitter plate presents the lowest average drag coefficient.

In the case of \( U_s = 20 \), where the flexible flag is deformed by the action of the downstream flow and a regular flapping state is present, we observe a reduction in averaged drag coefficient. However, the rigid case is still the case with the higher reduction.

The case where \( U_s = 30 \), the flexible flag presents the irregular flapping state, here also we observe the reduction in the average drag coefficient, but rigid presents the smaller averaged drag coefficient.

### 7.6 Summary chapter 7

In this chapter we have identified:

- During this chapter, we use two different methods to define the boundaries of both bodies. In the case of the flat plate, staircase approach is used and Immersed boundary method for
a flexible flag.

• We have proposed a set of experiments to analyse the performance of the flexible flag behind a flat plate. We considered three Reynolds numbers, three angles, four reduced velocities and four mass ratios.

• A set of experiments to test the Linear Stability Analysis were selected for three different Reynolds numbers. The primary goal of this experiments was to identify the limits of different states presented previously in the literature.

• Once that limits were identified, ranges of reduced velocity and mass ratio were selected to avoid irregular flapping state. The selected reduced velocity was $U_s=5-30$ and mass ratio $\mu =0.05-0.3$.

• Averaged tail displacement frequency was calculated to identify points where the flexible flag presented a significant displacement.

• When angle $\alpha=40$, the flexible flag presents several possible performances, for that reason we select this angle and two Reynolds numbers to observe in detail the performance of the flexible flag.

• A tail displacement analysis was done; also we simulated the cases of just a flat plate and a flat plate with a rigid flag, which is acting as a splitter plate.

• The tail displacement presented similar results for both cases, no matter the Reynolds number. Firstly when reduced velocity $U_s = 5$, we observe that flexible flag displacement uniformly, with a constant velocity and single frequency. The performance is confirmed with instantaneous vorticity figure, where we find that flexible flag displaces without losing its shape. This condition act as a moving splitter plate.

• When reduced velocity $U_s=10$, we observe that flexible flag does not show any displacement, velocity or frequency. When we observe the instantaneous vorticity, the flexible flag presents an initial, but later it remains trapped by vortex shedding produced by the flat plate.

• As soon as we increase the reduced velocity $U_s=20$ the tail displacement analysis shows a uniform displacement, constant velocity and single frequency but the difference appears when we check the instantaneous vorticity is that flexible flag deforms by the action of downstream flow.

• When $U_s=30$, we start to observe an irregular displacement with snapping events and more than one frequency. Here, the flexible flag displaces irregularly, and it loses its shape.

• The dynamical states found in these simulations are: I) A regular flapping state were the flexible flag displace uniformly, and it does not change its shape. II) A state where the
flexible flag remains trapped by vortex shedding. III) A regular flapping state, where the flexible flag displace uniformly, but it loses its shape. IV) An irregular flapping state, where the flexible flag loses its form.

• Pressure coefficient was measured as well. The rigid plate was the best option to reduce pressure on the rear face of the flat plate.

• All cases presented shown a drag reduction in comparative just the flat plate, however when flag acts as splitter plate a higher amount of drag is reduced.
In this work we have studied the performance of a flexible flag attached to a flat plate. We considered three different Reynolds numbers, three incidence angles, four reduced velocities and four mass ratios.

The literature review shows us that the splitter plates, rotating splitter plates and flexible bodies are an alternative to reduce drag and suppress vortex shedding. In this sense, there are several possible approaches to investigate this phenomenon: experimental, theoretical and numerical simulations. In the case of numerical simulations, we observe that Immersed boundary method has been a technique widely used to simulate the performance of a flexible flag, while fluid flow is simulated through the Navier-Stokes equations or the Lattice Boltzmann methodology. Lattice Boltzmann method has shown that it is suitable to simulate this phenomenon due to it is highly parallelizable, it has presented accuracy to treat simple, complex or deformable geometries.

The literature also shows that three main parameters can control the flexible flag performance: mass ratio, bending rigidity and Reynolds number. Previous studies have focused their attention in measure flapping frequency, flapping amplitude, tail displacement and trajectory, vortex shedding and flapping pattern as well as arrays of flexible flags. Some papers have focused their attention on studying a flexible flag attached to a bluff body, but most of them use symmetrical shapes as cylinders. There is a gap in this kind of studies but asymmetrical shapes.

In this work, we proposed a set of experiments to understand the performance of a flexible flag attached to a flat plate due to the lack of information for this phenomenon in literature. To study the performance of a flag attached to a flat plate, we developed an entirely new code based on Lattice Boltzmann method to simulate fluid flow and immersed boundary method for deformable bodies; this code is called LUMA. LUMA is parallelised through Message Passing Interface, and it is possible to use the refinement mesh methodology.

In the experiments shown in this work, we couple for the first time two different types
of boundary conditions to define the immersed body. We couple two boundary conditions, the staircase approach for a solid body and the Immersed boundary method for the flexible body. The staircase approach is suitable for the simple object, for example, the flat plate while Immersed boundary method can handle with deformation, in this case, the flexible flag.

Before to propose the experiments of a flexible flag attached to a flexible body, we reproduced a set of tests previously published in the literature. To measure parameters as drag and lift coefficient, Strouhal number, pressure coefficient and time average separation bubble we follow the experiments proposed by Park [75] having excellent performance. We changed the bluff body for a flat plate, and we reproduced the Taira and Colonious [84] experiments. During these experiments, the incidence angle was modified, and Reynolds number remains constant. Averaged drag and lift coefficient were compared and LUMA code reproduce appropriate results.

As well as the case of the bluff body, we select a set of experiment to test LUMA for a flexible body, Lee and Huang [52]. We developed a tail displacement analysis. We measured tail displacement, tail velocity and frequency analysis. For mass ratio $\mu=0.1$, the tail remains parallel to the flow. If mass ratio $\mu$ increases the displacement of the tail increases, for mass ratio $\mu=0.1$, 0.2, 0.8, and 2.0 we obtained similar results as the reference work, but for mass ratio $\mu=3.0$ despite we obtain same irregular performance the displacement is different. Here, we reproduced the three states previously reported in the literature, I) A straight-state, where flexible flag remains parallel to the flow, II) a regular flapping state, where flexible flag displace uniformly with constant velocity and an III)irregular flapping state, where flexible flag displaces irregularly with snapping events. We performed a bifurcation behaviour analysis as well. We tested several combinations of mass ratio, reduced velocity and Reynolds number. The model implemented in LUMA overpredicts the limits in comparative with the Linear Stability Analysis presented by Connell [27], but as well as other models follow similar patterns.

Once that we obtain an appropriate result for different bluff bodies and a flexible flag, we proposed a set of experiments to study the performance of a flexible flag attached to a flat plate. We followed the next steps

- A Linear Stability analysis was done for three different Reynolds numbers $Re=100,200$ and 400 to identify the limits where a flexible flag could perform with an irregular flapping state.

- We identified ranges of mass ratios ($U_s = 5 – 30$) and reduced velocities ($\mu = 0.05 – 0.3$) where the flexible flag will response with a straight-state or a regular flapping state.

- An analysis of average flapping frequency was done to identify the parameter values where flexible flag register displacement.

- A tail displacement analysis was done to understand the performance of the flexible flag.
We found that when angle $\alpha=40$ and Reynolds $Re=200,400$ the flexible flag shows a performance did not mention previously in the literature. Once that we analyse the tail displacement, we were able to identify four different states I) A regular flapping state where the flexible flag displace uniformly, and it does not modify its shape, it is acting as a rotating splitter plate. II) A state where the flexible flag remains trapped by vortex shedding, it registers an initial displacement but later remains in a single position. III) A regular flapping state, where the flexible flag displace uniformly, but it loses its shape. IV) An irregular flapping state, where the flexible flag loses its shape.

Pressure coefficient was measured as well, and the rigid flat plate presented the highest reduction of pressure on rear face followed by the flexible flag with $U_s = 5$. Flexible flag $U_s = 20$ and 30 with cases show similar pressure reduction on the rear face. This performance can be confirmed with drag coefficient where we observed that the rigid case presented a higher reduction in the average drag coefficient in comparative with the flexible flag cases.

In this work:

- We create an entirely new code to simulate the performance of a flexible flag attached to an asymmetrical bluff body.

- We coupled two different boundary conditions for the bluff body and the flexible flag.

- We identify four different dynamical states that have not been previously reported in the literature.

- A rigid splitter plate is still the configuration with higher drag reduction. However, some possible modifications, for example, flag length, could modify this performance.

In the next section we will talk about future work and some question that we need to address.
8.1 Future Work

The study of a flexible flag behind a bluff body has shown to be an interesting topic due to the number of parameters that we can be modified.

We still need to look at our attention to:

- Most of the presented experiments were done a low mass ratios and moderately reduced velocities, we still need to explore higher mass ratio and reduced velocities, mainly in the regular flapping state.
- The incidence angle has shown to play an important role due to in many cases at lower angles we do not capture any displacement, we still need to explore higher angles
- A deep study of the four dynamical states identified during our experiments.
- A bifurcation behaviour map can be built at least for one Reynolds number.
- The experiments presented in this work were done for a flag of the same length, we still need to modify the length of the flag
- A different bluff body can be used, for example, an airfoil.
- Develop a three-dimensional flag model. Some experiments for three-dimensional flat plate with different incidence angles have been done with similar results as reference.
- We still have to test a flat plate for an array of flags.
A.1 Introduction

In this appendix, we are going to describe a high level introduction to Message passing interface. Firstly, we are going to comment about MPI description and history. Secondly, we are going to describe the process of coupling Lattice Boltzmann Method and Message passing interface. To finish we are going to explain the refinement levels process.

A.2 A brief history of MPI

Message passing interface (MPI) [3] is a communication system that has been designed to program parallel computers over a distributed memory in Fortran, C or C++. MPI communications can be point to point or collective and it was designed to:

- Practical.
- Portable.
- Efficient.
- Flexible.

MPI was created between the late 80’s early 90’s due to the need to have an standard memory distribution, until this point Parallel and fix computing were incompatible and MPI was developed to address issues previously mentioned.

Therefore organizations, software vendors and writers as well as scientists now posses an standardization of MPI libraries, there are several versions of each one containing different features. Table A.2 shows a summary of the different implementations and the common uses.
APPENDIX A

APPENDIX A

Year   MPI history
1992   MPI standardization is established and a draft is proposed.
1992   MPI procedures are accepted by organizations involved in software development.
1993   MPI draft standard were presented.
1994   MPI 1.0 version was presented.
1998   MPI 2.0 was presented.
2012   MPI 3.0 is approved.

<table>
<thead>
<tr>
<th>Year</th>
<th>MPI use</th>
</tr>
</thead>
<tbody>
<tr>
<td>MPICH</td>
<td>Linux clusters</td>
</tr>
<tr>
<td>Open MPI</td>
<td>Linux clusters</td>
</tr>
<tr>
<td>Intel MPI</td>
<td>Linux clusters</td>
</tr>
<tr>
<td>IBM BG/Q MPI</td>
<td>BG/Q MPI</td>
</tr>
<tr>
<td>IBM spectrum MPI</td>
<td>Coral early access clusters</td>
</tr>
</tbody>
</table>

Table A.1: MPI history.

Table A.2: MPI Implementations

Up until this point, MPI history as well as the different implementations have been roughly summarized, now it is time to present the MPI program structure and nomenclature.

A.2.1 MPI program structure

The MPI structure can be seen in figure A.1. As we can see, any software based on MPI can be divided into two main sections: Serial and Parallel. The Serial section keeps all segments where there is a certain dependency on the instructions running among other cores, while Parallel is used for independent instructions that can be run at same time, as they do not depend on other calculations.

However in order to have effective communication among processes, we have to take into account some concepts used by MPI developers to define the type of communications to map the process. Table A.5 presents some important concepts:

Here, we have shown some MPI communicators that are used to identify operations as well as the calculated time duration by each process. The main role of the communicator objects is to connect groups, give identity and order in a topology of all processes. In the next section we will define the MPI communications.

A.2.2 Communication modes: Point to point or collective communications

MPI communications can be divided into two main modes of Point to point and Collective. The main purpose of Point to point is to transfer a direct message between specific processes. This communication requires from the action of the sender and receiver [4].

Point to point can be blocking and non blocking communication:
A.2. A BRIEF HISTORY OF MPI

Figure A.1: MPI program structure.

<table>
<thead>
<tr>
<th>MPI Routines</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>MPI_COMM_WORLD</td>
<td>Predefined communicator.</td>
</tr>
<tr>
<td>RANK</td>
<td>Process identifier.</td>
</tr>
<tr>
<td>MPI_Init</td>
<td>Initializes MPI environment</td>
</tr>
<tr>
<td>MPI_Comm_size</td>
<td>Obtains the number of ranks</td>
</tr>
<tr>
<td>MPI_Comm_rank</td>
<td>Obtains the rank of specific communicator</td>
</tr>
<tr>
<td>MPI_Abort</td>
<td>Finishes MPI process</td>
</tr>
<tr>
<td>MPI_Get_processor_name</td>
<td>Returns processor name</td>
</tr>
<tr>
<td>MPI_Get_version</td>
<td>Returns MPI version</td>
</tr>
<tr>
<td>MPI(Initialized)</td>
<td>Asks whether MPI Init has started</td>
</tr>
<tr>
<td>MPI_Wtime</td>
<td>Returns an elapsed wall clock time in seconds</td>
</tr>
<tr>
<td>MPI_Wtick</td>
<td>Returns resolution in seconds</td>
</tr>
<tr>
<td>MPI_Finalize</td>
<td>Finish with MPI environment</td>
</tr>
</tbody>
</table>

Table A.3: MPI Communicators
APPENDIX A

<table>
<thead>
<tr>
<th>Point to point Communication</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>MPI_send</code></td>
<td>Blocking</td>
</tr>
<tr>
<td><code>MPI_recv</code></td>
<td>Blocking</td>
</tr>
<tr>
<td><code>MPI_Ssend</code></td>
<td>Blocking</td>
</tr>
<tr>
<td><code>MPI_Sendrecv</code></td>
<td>Blocking</td>
</tr>
<tr>
<td><code>MPI_Isend</code></td>
<td>Non-blocking</td>
</tr>
<tr>
<td><code>MPI_Irecv</code></td>
<td>Non-blocking</td>
</tr>
<tr>
<td><code>MPI_Issend</code></td>
<td>Non-blocking</td>
</tr>
</tbody>
</table>

Table A.4: MPI Communicators

<table>
<thead>
<tr>
<th>Collective</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>MPI_Barrier</code></td>
<td>Synchronization operation.</td>
</tr>
<tr>
<td><code>MPI_Bcast</code></td>
<td>Data movement, same message from one rank to all processes</td>
</tr>
<tr>
<td><code>MPI_Sscatter</code></td>
<td>Data movement, different message from one rank to all processes</td>
</tr>
<tr>
<td><code>MPI_Gather</code></td>
<td>Data movement, different message from each rank to a single destination</td>
</tr>
<tr>
<td><code>MPI_Allgather</code></td>
<td>Concatenation of data to all tasks in a group</td>
</tr>
</tbody>
</table>

Table A.5: MPI Communicators

- The blocking process will stop and will not continue until the message arrives to specific process to ensure that the message data received.
- The non-blocking communication does not need to stop to ensure that the message has been received. The process just requests to start the operation and then continues processing.

We can summarized point to point communications routines in table A.5. However as previously mentioned, there are several other options to Point to point communications that are used in this this work. In the case of Collective communication message is taken and sent it to all processes. Table A.5 presents a summary of collective communications.

A.3 Lattice Boltzmann Method and MPI

As mentioned in previous chapters, the Lattice Boltzmann Method is a highly parallel computational approach and due to the amount of lattice points in our research, it is necessary to develop an strategy to decompose domain to perform paralellization.

The domain decomposition method consists of sending and receiving data among processors, which are sub-domains of the main computational domain. Data is shared through the halo cells, which act as inner boundaries. There are several alternatives to communicating messages through MPI: blocking, non blocking and combining, appendix A. On the one hand a blocking communication does not return until communication is finished. On the other hand, non-blocking communication is made even if communication is not finished.
A.3. LATTICE BOLTZMANN METHOD AND MPI

In the next section we will talk about domain decomposition.

A.3.1 Lattice Boltzmann Method and MPI decomposition

In this section, we are going to describe the implementation process of the Lattice Boltzmann methodology on MPI. We can found some documents that implemented MPI for Lattice Boltzmann theory, for example Davison [28] or Brair [21]. The process consist in four steps:

1. The domain should be broken up into blocks.
   - The computational domain must be divided among processors; these adjacent sub-domains share interfaces the interface cells are known as halo or ghost cells.

   ![Figure A.2: Split domain into blocks.](image)

2. These blocks should assign MPI processes one by one.
   - All sub-domains should receive a number.

   ![Figure A.3: MPI assignation.](image)

   - Each grid must have a map of the neighbour's processes. Halos hold copies of shared interfaces; these values are going to be updated when original values change.

3. Apply stream step and swap information.
   - As can be seen on figure 3.9, the Lattice Boltzmann methodology has 2 main steps; collision and stream. Collision is a local step; it is not required to share information with neighbouring cells. However, propagation or streaming step requires communication with neighbouring cells and here is where it is required to identify which process will receive information from neighbouring processes.
4. Apply boundary conditions.

   • During this step, information is passed to proper neighbouring processors, halo cells are updated and boundary conditions are applied as normal.

Up until this point, we have described the process to couple Lattice Boltzmann Method and Message passing interface, however we have to explain the refinement levels methodology in order to have a finest resolution close to immersed body.

A.4 Refinement Region method

In the Lattice Boltzmann method, a square or cubic lattice grids is commonly used in order to avoid any kind of interpolation; therefore there is no need to introduce an extra numerical viscosity. However, it is common to require high resolution meshes around irregular geometries as well as capture the smallest scale effects in order to obtain more accurate results. The easiest solution is to increase the resolutions of all computational domains. However, this is not a clever solution and impacts directly on computational time. LUMA uses an embedded uniform grid developed by Rohde [79] where mass conservation is imposed but during the propagation steps allows particles to move from the coarse region to the finest region and vice versa.
A.4. REFINEMENT REGION METHOD

A.4.1 Local embedded uniform grid

In the paragraph, we are going to explain the process followed by Rohde [79]. Figure A.6 represents a typical computational grid, which consist of a set of local embedded uniform grids, $\Delta x_c$ and $\Delta x_f$ are the lattice space for coarse and fine regions respectively. Furthermore, figure A.6 $P_1$ represents the grid node at coarse region and $P_2$ is the node at the finest one.

Before the explanation, we have to clarify that cells marked with $A$ represent the coarse grid cell and $B$, the finest region cells. The circles $\bigcirc$ and squares $\blacksquare$ represent the center of the coarse and fine grid. In the same way, open arrows ($\wedge$, $\vee$) correspond to particle densities that move from fine grids and closed arrows ($\triangle$, $\triangledown$). Particle densities are originate from coarse grid cells. Particle distributions after a streaming step are represented by arrows pointing towards the cell center; particle distributions after a collision step are denoted by arrows pointing from the cell center.

The methodology introduced by Rhode starts with:

1. Collision step on coarse and fine grid cells.
   
   - Collision step and body force are applied from coarse to finest regions.

\begin{equation}
    n_i(x,t^*) = n_i(x,t) + \Omega_i(N) + t_f, i\rho(c_i \cdot G) \quad (A.1)
\end{equation}

Figure A.7: Collision step coarse and fine region.
APPENDIX A. APPENDIX A

Where \( t_* \) is the moment after collision and last term is the addition of momentum to the fluid due to the presence of a body force \( G \).

2. Homogeneous redistribution of particle densities from coarse to fine grid cells.

![Homogeneous redistribution from coarse to finest regions.](image)

\[
(N_i(x_p,t_*))_f = \frac{1}{n^D} (N_i(x_c,t_*))_c \text{ with } \begin{cases} 
  p = 1, \ldots, n^D \\
  x_p = x_c \pm \frac{1+2k}{2n} \Delta x_c \\
  \text{with } k = 0, \ldots, n^D - 1 
\end{cases} \tag{A.2}
\]

Where \( N_i = \Delta V \) is the volume of a grid cell. Equation A.2 represent the particle densities in the smaller cubic cells, which are going to be equal to the original particle density in the coarse regions; hence \((n_i)_f = (n_i)_c\). This methodology does not apply any spatial interpolation or rescaled particle densities.

3. Streaming step on the coarse and fine grid.

![Streaming step coarse and fine grids.](image)

- Apply propagation step in coarse and fine grids.
• Both grids start communication; coarse region remains unchanged and propagates to the finest grid. Collision operations was previously applied.

4. Repeat steps 4a and 4b \( n-1 \) times.

• Collision step on the fine grid 4a.

- Particles that have already been propagated from the fine to coarse region will not participate at any collision step again.
- Particle densities on \( \tilde{t} = \tilde{t}_0 + \Delta \tilde{t} \) which are located at share nodes \( P_1 \) and \( P_2 \) are going to be obtained by interpolating the rescaled coarse distributions on \( \tilde{t} \) and \( \tilde{t}_0 + \Delta \tilde{t} \).

• Streaming step on the coarse and fine grid.

5. Homogeneous redistribution of particle density from fine to coarse grid cells.

• All \( C_i \) pointing in the coarse grid direction must be the sum, resulting in a new incoming particle distribution function.

\[
(N_i(x, t))_c = \sum_{p=1}^{n_p} (N_i(x_p, t))_f
\]  

(A.3)
or in terms of particles densities:

\[
(n_i(x,t))_c = \frac{1}{n} \sum_{p=1}^{n_p} (n_i(x_p,t))_f
\]  \hspace{1cm} (A.4)

Up until this point we have explained the process to apply local embedded uniform grid method.

### A.5 Summary appendix A

In this appendix we have identified:

- We have explained the process to couple lattice Boltzmann method with message passing interface.

- We explain the process of embedded grid refinement and how it work on Lattice Boltzmann Method.

[2] InnovativeCFD.

https://computing.llnl.gov/tutorials/mpi/.

http://mpitutorial.com/tutorials/point-to-point-communication-application-random-walk/.


[6] OpenLB.
http://optilb.org/openlb/.

http://www.palabos.org/.


