

CRANFIELD UNIVERSITY

Evandro Liani

**Potential Flow Based Aerodynamic and
Aeroelastic Analysis of Flapping Wings**

SCHOOL OF ENGINEERING

PhD THESIS

CRANFIELD UNIVERSITY

SCHOOL OF ENGINEERING

PhD THESIS

Academic Year 2007-08

Evandro Liani

Potential Flow Based Aerodynamic and Aeroelastic Analysis
of Flapping Wings

Supervisor: Dr Shijun Guo

March 2008

This thesis is submitted in partial fulfillment of the requirements for the degree of
Doctor of Philosophy

©Cranfield University 2008. All rights reserved. No part of this publication may
be reproduced without the written permission of the copyright owner.

To My Family...

Abstract

The motivation for this research is the importance that the modelling of aerodynamics and aeroelasticity of flapping wings has had in the last decade. The development of flapping wings Micro Air Vehicles (MAV) has captured a huge interest in the recent past, due to the several disciplines involved in the subject. In this dissertation the attention is focused on the flow and its interaction with the structure.

Even though experiments have had a fundamental role in the explanation of the aerodynamics around a flapping wing, it is widely accepted that a key aspect in the development of future flapping wings MAVs is the modelling. The aim of the project is to investigate different techniques for the development of a numerical framework used in the prediction of the unsteady aerodynamic forces on flapping wings.

The understanding of the phenomena occurring on flapping wings is attempted first with very basic models. The research is carried out based on potential flow assumptions: the flow is initially treated as irrotational and inviscid. Although the assumptions are very strong, it is shown that the mechanism of lift and thrust production can be described, together with the convection of the wake behind the wing.

The limitation of potential flow models is the incapacity to describe flows that are separated over a large portion of the wing. The modelling of this issue is particularly important for flapping wings, where the separation is exploited in order to increase the forces produced.

The development of a Vortex Particle Method (VPM) is attempted, with wake elements released at each time step from all the panels of the airfoil. The advantage of Vortex Particle Methods over panel methods is that they represent more realistically the flow around the wing. The drawback is the greater complexity and longer running times.

Aeroelasticity is discussed as well, as it is believed that the wing flexibility can enhance the performance of flapping wings. The thesis investigates the stability and response of an airfoil connected to a rotational and a linear spring at its elastic axis. Even though the structural model is very simple, it is shown that there might be advantages introducing a certain level of flexibility in the system.

The framework built in this project is not aimed at giving an accurate representation of the forces produced by flapping wings. A methodology that allows to avoid CFD computations is deemed fundamental in the design phase of an aerial vehicle. The final goal of this project is the development of meshless techniques for the aerodynamic analysis of unsteady flows. An essential point that needs deep insight is their inaccuracy compared to CFD. Therefore considerations about the lack of accuracy and ways to improve it are made in order to show that there is a real advantage in the use of grid-free methods. The results

of the analysis are compared with other results found in the literature. In particular, experimental results are considered where possible, otherwise numerical computations have been taken into account. The first part of the code has been developed in FORTRAN, due to its running time efficiency, while the second part has been developed in C++, because of its ability to handle more complex data structures.

Acknowledgements

The last three years spent at Cranfield have been among the best years of my life so far. With the following lines I wish to express all my gratitude to those people that contributed to this.

I certainly owe a debt of gratitude to my PhD advisor, Dr Shijun Guo. The guide and the support he provided as a supervisor have been invaluable. I also feel obliged towards Dr Giuliano Allegri for the time spent to help me overcoming the hurdles of my research project.

The greatest gratitude goes to my family, for their endless patience and support during all these years of study. They have always made my life easier, allowing me to look after minor issues only. I wish to let them know that their words of encouragement have always been listened to. Their long term investment is finally paying off.

A few words must be spent for the wonderful environment in which I have been working at Cranfield. I am talking about the people I met here, whose friendship has meant a lot to me during this journey. I am thankful to Giovanni, my housemate, and to all those friends and colleagues whose presence has made Cranfield a better place to live. I

wish to them the best of luck for the future.

There is a person I feel my last Thanks should go to. She is the person I started a journey with right before the PhD started. We shared the good and the bad moments of each other's lives, but we never had the chance to live our relationship like a normal couple. These few lines stand for all the words I should have said and I didn't . Thanks a lot, my love.

Evandro.

Contents

Contents	vi
1 Introductory Remarks	1
1.1 Layout of the Dissertation	3
2 Background	9
2.1 Flapping Wing Micro Air Vehicles	10
2.2 Flapping Wing Aerodynamics	14
2.2.1 Aerodynamics: Understanding of the flowfield .	17
2.2.2 Potential Flow Based Aerodynamics	20
2.2.3 Panel Methods	26
2.2.4 Vortex Methods	34
2.2.5 CFD Methods	48
2.3 Aerodynamics:Experimental Results	51

2.4	Aeroelasticity of Flapping Wings	55
3	Methodology and Analysis	60
3.1	Theodorsen Theory	60
3.2	Unsteady Aerodynamics of Flat Plate Undergoing Large Amplitude Oscillations	65
3.3	Panel Methods	71
3.4	Extension of Panel Methods to Multiple Airfoils	81
3.5	Panel Method Implementation	83
3.6	Vortex Particle Methods	97
3.6.1	VPM:Implementation Details	102
3.7	Aeroelasticity of FW	119
4	Results and Discussions	136
4.1	Deformable Wavy Wake	137
4.2	Single Airfoil Flapping	147
4.3	Multiple Airfoil Flapping	161
4.4	Vortex Particle Method	168
4.5	Aeroelasticity of Flapping Wings	187
4.5.1	Aeroelastic Response of Flapping Wings	193

5	Conclusions and Recommendations	197
	References	201

Nomenclature

f = frequency of oscillation (Hz)

c = chord length

b = half chord length, $\frac{c}{2}$

U_∞ = Free stream speed

a = Non dimensional distance between elastic centre and the centre of the airfoil, positive backwards

k = Reduced Frequency, $2\pi fb/U_\infty$

C_p = Pressure Coefficient on the airfoil surface

C_L = Lift coefficient per unit span

C_M = Moment coefficient per unit span

C_T = Thrust coefficient per unit span

C_P = Power Coefficient

η = propulsive efficiency, $\frac{C_T}{C_P}$

L = Lift per unit span

M = Pitching Moment per unit span

T = Thrust per unit span

$C(k)$ = Theodorsen function, $F(k) + j G(k)$

h = Vertical displacement of the wing section

α = Angle of Attack

Δt = Time step

σ = Source Strength distribution

μ = Doublet Strength distribution

γ = Vortex strength density over the panel

Γ = Strength of wake vortices

ϕ = Perturbation potential

Φ = Total Potential, equal to undisturbed flow + perturbation potentials

ρ = density of air

ν = Dynamic viscosity

Re = Reynolds number, $\frac{U_\infty c}{\nu}$

St = Strouhal number, $\frac{\omega h}{U_\infty}$ (for plunge motion)

x, z = Coordinates of a point in the flowfield

x_1, z_1 = Origin of the panel

x_2, z_2 = Right vertex of the panel

v_τ, Q = Velocity tangential to the generic body panel

$v^{x,z}$ = Components of velocity induced on wake vortices

$v_b^{x,z}$ = Components of velocity of the airfoil

$b_{x,z}$ = Unitary influence of body sources on the body

$m_{x,z}$ = Unitary influence of body doublets on the body

$w_{x,z}$ = Unitary influence of wake doublets on the body

$b_W^{x,z}$ = Unitary influence of body sources on the wake

$m_W^{x,z}$ = Unitary influence of body doublets on the wake

$w_W^{x,z}$ = Unitary influence of wake doublets on the wake

$\mathbf{n}$ = Vector normal to the body panel, calculated in the middle of the panel

N_P = Number of panels on the body

N_W = Number of time steps and of vortices released in the wake

ζ_ϵ = Mollification kernel tending to dirac distribution as $\epsilon \rightarrow 0$

α_p = Strength of vortex particle in VPM

ω = Curl of the velocity field, $\nabla \times u$

$\delta(x)$ = Dirac Function

Symbols specific to Aeroelasticity:

$\mathbf{M}, \mathbf{K}, \mathbf{E}$ = Mass, Stiffness and GAF Matrix

$\mathbf{q}$ = Generalized Lagrange Coordinates

s, p = dimensional and non dimensional poles of the aeroelastic system

$\lambda, \mathbf{v}$ = Eigenvalues and eigenvectors of the 2×2 system

$\mu, \mathbf{Z}$ = Eigenvalues and eigenvectors of the 4×4 system

$\mathbf{A}_0, \mathbf{A}_1, \mathbf{A}_2, \mathbf{P}, \mathbf{R}$ = Finite state aerodynamic matrices

Ω = Diagonalized stiffness matrix

q_D = Dynamic Pressure

List of Acronyms

MAV = Micro Air Vehicle

FW = Flapping Wing

UTIAS = Potential flow model for flat plates (De Laurier-Fairgrieve)

FWW = Frozen Wavy Wake

DWW = Deformable Wavy Wake

PM = Panel Method

UPM = Unsteady Panel Method

VPM = Vortex Particle Method

PSE = Particle Strength Exchange

RWM = Random Walk Method

DVM = Discrete Vortex Method

FMM = Fast Multipole Method

DOF = Degrees of Freedom

RHS = right hand side

NSTEPS = Number of time steps used for the simulation

NCYCLES = Number of complete oscillations

NSAMPLES = Number of time steps for each cycle

GAF = Generalized Aerodynamic Forces (Matrix)

Chapter 1

Introductory Remarks

The reason for which the design of Micro Air Vehicles (MAVs) is studied is the importance of MAVs for both military and civil applications. Given their dimension, below $15cm$ wingspan, and their weight, maximum $100grams$, the fields in which they can be used are innumerable. To mention a few, their use has been thought to be particularly extensive in surveillance and reconnaissance missions, where they would carry a camera to film an environment beyond an obstacle without sending human personnel. Due to the high maneuverability, they can also be flown in tight spaces, for rescue missions in case of fire or within pipes where the access to rescuers would be impossible. A recent paper by Pines and Bohorquez [1] has reviewed the progress that in the last 15 years has been made in the development of these flying machines. Given their size and the speed at which operations are thought, they can be directly compared to insects and birds. The evolution has led these species to improve the mechanism for lift and thrust production. Therefore to mimic the wing motion of animal species has become the

target of many researchers investigating flapping wing propulsion.

Another paper by Woods, Henderson and Lock [2] shows that, for the combination of cruising speed and weight of interest for Micro Air Vehicles, flapping wing is the most efficient means of propulsion. The conclusion of the above mentioned paper is based on analytical theories that yield the energy required for flight without exploring the ways in which the energy is going to be produced. In large planes, the source of the thrust is the engine, whereas the lift is provided by fixed wings. Scaling to the size of micro air vehicles, the Reynolds number, defined as

$$Re = \frac{U_{\infty} c}{\nu}$$

plays a relevant role in the description of the flowfield. The steady and quasi steady aerodynamics assumptions fail to explain how insects and birds can produce forces as large as ten times their weight. Experimental results have been obtained and they can partially explain the mechanism of forces production, but take a long time and cannot be used for design purpose. Therefore it becomes fundamental to develop tools able to represent the flowfield around a flapping wing at low Reynolds number.

It would be arrogant to pretend that the aerodynamics is the key-point that needs a solution in order to build flapping wing MAVs. In fact, as Pines [1] explains in his review paper, "*...if MAVs are to approach and*

possibly exceed the performance of biological flyers, advances are required in several fundamental areas including 1. low Reynolds number aerodynamics, 2. lightweight and adaptive wing structures, 3. energy storage/conversion to useful power/ propulsion, and 4. insect flight navigation, guidance and control.” The present dissertation is focused on the investigation of flapping wing aerodynamics, as one of the key points to be investigated for the efficient design of these new kinds of air vehicles.

1.1 Layout of the Dissertation

The main body of the thesis is focused on the development of a computer code for the efficient and accurate representation of the flow field around an airfoil undergoing plunge and pitch oscillations in the plane in which the airfoil is lying. The length of the report may confuse the reader on the objective of the work, therefore it seems fair to provide the reader with an overview of the subjects treated in the following chapters. Keeping in mind the aim of the present work, the layout has been thought in such a way that each subject results as the development of the previous one. The reasoning that has brought to such developments is explained where it is reckoned necessary for the sake of understanding.

Following these introductory remarks, **chapter 2** is a review of the work that has been done previously on the design of micro air vehicles.

After a brief introduction on the concept of micro air vehicles and the different configurations designed in the past, the attention is shifted on the work that has been done on flapping wings micro air vehicles. It is emphasized the fact that flapping is the most efficient means of propulsion when particular requirements must be met. Therefore a motivation is given for the investigation of flapping wing aerodynamics. The discussion goes on with the description of the phenomena that are believed to be important for a correct representation of the flow field. For modelling purposes, it is essential to assess whether or not a phenomenon will be captured with the model being used. An important contribution to the development of the knowledge on the subject has been given by experimental results: although the work is mainly focused on the development of numerical tools, noteworthy experimental investigations are reported to provide the reader with a 360-degree review of the subject. The next steps is to describe the different options available to model the aerodynamics of flapping wings: in this thesis three of them are considered, and the motivation is given at this point. Last but not least, the subject of aeroelasticity is discussed. The advantage of embedding flexibility in the modelling is shown, as extra forces can be produced with a flexible wing.

Chapter 3 shows the development of the numerical methods used in the dissertation. The basis of the work is the theory of Theodorsen, explained in the first paragraph of the chapter. The importance of this model is to provide a benchmark to validate the numerical analysis carried out in the rest of the thesis. The theory also represents

the simplest model developed for flapping wing aerodynamic analysis, but it is still fundamental to understand the basics of force production in flapping wing flight. The assumptions of Theodorsen are removed one by one, adding complexity to the analysis up to the point where CFD is thought to be needed to yield a greater accuracy. The models presented are respectively: UTIAS, differing from Theodorsen because it removes the assumption of small oscillations; Unsteady Panel Method (UPM), where the airfoil computed can be of arbitrary shape and thickness; Vortex Particle Method (VPM), in which the assumption of attached flow is relieved.

Theodorsen and UTIAS models have been implemented into a FORTRAN code, but no further improvement has been brought to the theory. They are used as benchmarks to test more complex models against. The Unsteady Panel Method presented in the report has been developed based on the work of Jones and Platzer ([3, 4, 5, 6, 7, 8, 9, 10, 11]) and Katz and Plotkin ([12]). It has been initially built to compute the forces around an airfoil undergoing arbitrary pitch and plunge motion, and then adapted to compute the flowfield around two airfoils in biplane and Schmidt propeller configurations. The advantage of the panel method will be shown to be the running time, therefore it is attractive for the preliminary design phase of the aircraft. The drawback is that it cannot describe any kind of phenomena related to flow separation and the effect of viscosity. Therefore the next step in the analysis is the development of a potential-flow-based method which could describe the separation of the flow over the

airfoil's surface.

Included in chapter 3 is also the presentation of the method used for coupling the airflow and the airfoil's structure. Aeroelasticity is of great concern for maximising the performance of flapping wings. Therefore it has been considered in the present report, in its basics, to understand whether a real advantage can be gained from a flexible wing over a rigid one.

Chapter 4 presents the details on the implementation of the theory described in the previous chapter into a computer code and the subsequent validation of the code against data found in literature. The implementation of a model into a code is never straightforward, due to the finite precision of the computer, the choice of the algorithms, the accuracy needed and so on. Therefore it is believed that a detailed description of how the code has been implemented is necessary to save time to the people that will want to re-implement it. The attention is focused on those points to which the author has dedicated a long time debugging. In the case of the panel method and the vortex particle method, the point that takes the longest time is the convection of the vortex elements. Therefore particular attention will be given to how to achieve the desired accuracy without compromising the stability of the numerical model. The chapter also presents a validation of the model while it is being developed. It has been decided to do so because of the hurdles that the author has met during the development of the code. The availability of mid-results results allows, by experience, to

speed up the process of debugging and validation.

The results of the computations with the discussions are also given in **chapter 4**. The results obtained are compared with experimental data, where they were available, and numerical data found in literature. The results presented will consist mainly of the wake visualization behind the airfoil and the forces produced by the airfoil itself. The importance of taking the wake into account to represent the performance of flapping wings is demonstrated comparing the forces generated with and without the wake. If the convection process is "switched off" in the panel method analysis, the results for an airfoil that resembles a flat plate will be the same than obtained with Theodorsen. If the wake is switched back on, the contribution in terms of forces is visible in the plots. The effect of wake-airfoil interaction is investigated, in particular for the Schmidt propeller case and in the vortex particle method, in which each panel of the body releases a vortex. Moreover, a parametric study is made to assess the effect of the amplitude of the oscillation, the phase lag between pitch and plunge, the frequency of the oscillation and the Strouhal number, that accounts for a combined effect of frequency and amplitude. These parameters are all reckoned important in the performance of flapping flight. The results that are used for comparison are the thrust coefficient and the propulsive efficiency of the airfoil, whose values can be found in other numerical analysis in the literature.

The dissertation is concluded with recommendations for work to be

carried out in the future.

Chapter 2

Background

In this chapter the work that has been done in the past on the aerodynamics of flapping wing micro air vehicles will be described. By the definition of the Defence Advanced Research Projects Agency (DARPA), the size of micro air vehicles cannot exceed 15cm in length, width or height, with a maximum take off weight of about 100grams . MAVs should be capable of staying aloft for 20 – 60 minutes while carrying a payload of 20grams for a distance of 10km . Inspiration for the design of micro air vehicles has come from both large scale aircrafts and the nature. The idea is to shrink the conventional aircraft design down to the size of micro air vehicles. The problems that arise are mainly connected to the aerodynamic efficiency and the duration of the battery. Even with helicopter configurations it has been shown ([2]) that the efficiency decreases greatly when the size is too small. Therefore many researchers have attempted to reproduce the flight of birds and insects for the design of MAVs.

2.1 Flapping Wing Micro Air Vehicles

For a long time the mechanism by which some species of insects manage to fly remained unexplained. A combination of high frequency oscillations, control of the wake and extremely large efficiency figures allows them to hover and to perform very tight manoeuvres, besides sustaining forward flight for a long time. In a flapping wing MAV the main issues arising in the design process are:

- battery;
- materials;
- aerodynamics.

An efficient battery would allow the aircraft to stay aloft for enough time to accomplish the mission. Proper materials would allow to save energy and to flap the wings in a more efficient way. Finally, the aerodynamics needs to be investigated in order to approach the performance of insects. Jones and Platzer ([3, 4, 13, 14, 5, 6, 15, 7, 16, 17, 18, 8, 19, 9, 20, 10, 21, 11, 22, 23]) greatly contributed to the research in the field of Flapping Wing MAVs for their extensive study and design of a palm-size remote control flapping wing MAV.

Figure 2.1 shows the aircraft built by Jones and Platzer: the configuration is different from insect-like micro air vehicles, as the propulsive force is produced by two wings flapping vertically behind the main body, whereas the lift is produced by a fixed wing in front. The



Figure 2.1: MAV model built by K.D. Jones and M.F. Platzer ([10])

main advantage of this configuration is the mechanical stability: the symmetric flapping avoids the oscillations of the main body. Further advantages of this configuration are the simplicity of the motion, as plunge is the only active degree of freedom, and the prevention of separation on the main wing as a beneficial effect of the flapping wings. The configuration is also noteworthy because it exploits the **ground effect** caused by wings pair flapping in counterphase at very small distance. Some technical details of the MAV are given in the table below and have been taken from [11].

Model	Span	Length	Structural Weight	Total Weight
1	30 cm	20 cm	5.2 g	14.4 g
2	27 cm	19 cm	4.5 g	13.4 g
3	25 cm	18 cm	4.2 g	12.4 g
4	23 cm	17 cm	3.8 g	11.0 g

Table 2.1: Technical details of the MAV designed at Naval Postgraduate School

As shown in table 2.1, the structural weight is only one third of the total weight, two thirds being taken by the avionics, battery and motor.

The present model is capable of sustaining forward flight for 15-20 minutes, with a speed ranging from 2 to 5 m/s. As pointed out in [11], the efficiency of the flapping motion may be improved allowing the wings to have a passive pitch degree of freedom, increasing in this manner the forces produced.

A second prototype of flapping wing aircraft, far larger than a conventional micro air vehicle, was designed and built thanks to the work done at the University of Toronto's Institute for Aerospace Studies ([24]). Shown in figure 2.2, an engine-powered remotely controlled model was flown in 1991 for 2 min 46 sec. Improvement of that model led to the design of a human-carrying ornithopter, with the main characteristics given in [24]. Even though the aircraft cannot be included in the micro air vehicles family, it is an example of flapping wing aircraft, with an oscillation frequency of about $1Hz$ and a flapping amplitude of $31deg$. The frequency of oscillations is well below the range of interest for flapping wing MAVs, but the large amplitude of oscillations gives rise to such phenomena as stall on the wing, that are very important for the performance of insect-like flapping motions. Overall, the prototype built is also very useful for the amount of experimental data that it provides for comparison with numerical results.

Figure 2.3, taken from [1], presents further prototypes of flapping wings and other kinds of MAVs, plotted on a graph with the weight on the vertical axis and the mission endurance on the abscissa. The Aerovironment Microbat, developed in Caltech, is also a flapping wing

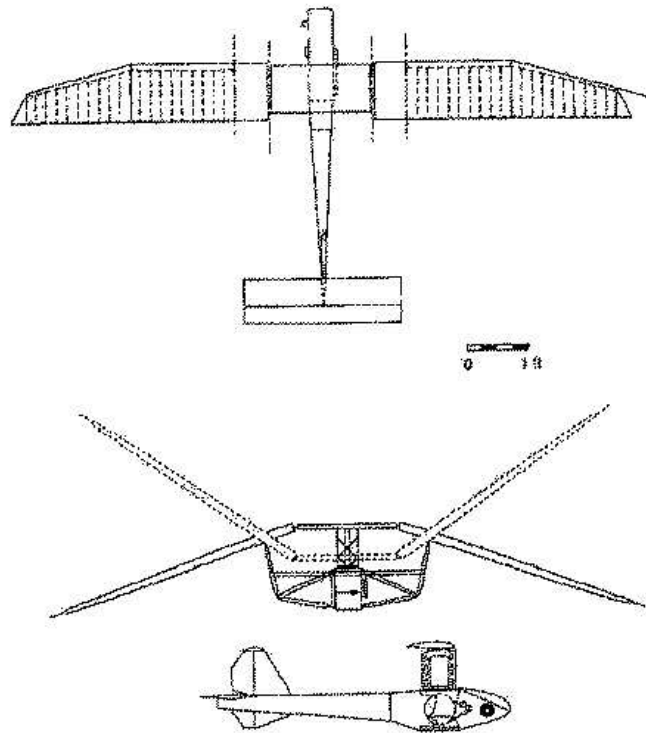


Figure 2.2: Different views of the flapping wing ornithopter [24]

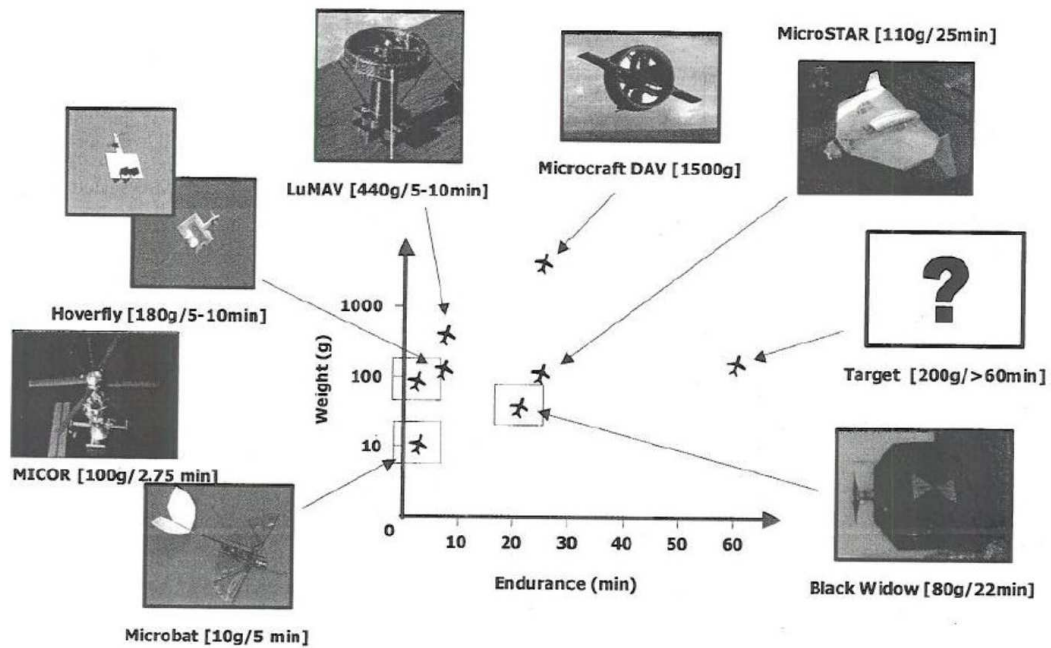


Figure 2.3: Prototypes of MAVs

MAV. It has a very light structure but also a very low endurance. The aircrafts that have the longest endurance are the Black Widow, developed in Aerovironment, capable of 22 minutes flight, and the MicroStar from Lockheed-Martin, capable of sustaining flight for 25 minutes. In both cases the thrust is provided via a front propeller, whereas the fixed wing provides the lift.

2.2 Flapping Wing Aerodynamics

As it has been outlined, the design of a Flapping Wing MAV requires the integration of different disciplines, among which are aerodynamics, structures, materials and flight control. The attention of this report is concentrated on the study of the airflow and its interaction with the structure. Due to the unsteady motion of the wings, the study of the flowfield is particularly complex. Therefore, an appropriate literature review is necessary to understand the nature of the basic phenomena. The proper understanding of the mechanism by means of which insect-like wings generate the lift will be followed by an assessment of the methods that can correctly describe the flowfield. It is worth pointing out that the literature review carried out in this chapter mainly concerns the development of numerical methods, but it does not provide a deep coverage of experimental work. Some experimental investigations will be presented for the sake of completeness, but that will be a limited part of the chapter.

In the Inaugural Goldstein Memorial Lecture in 1989, Sir James Lighthill gave an overview of what had been understood until that time about the flight of birds. The knowledge on this topic was essentially due to the work of *"certain very gifted biologists"* that *"have shown themselves able to apply most effectively these mathematical methods to challenging problems of notable biological interest"*. Most of the work that had been done up to that time on flapping wing flight came from numerical experiments carried out by biologists such as J. Rayner ([25, 26]) and G. Spedding ([27]). Similarities are observed between the results found by the authors above and the theory developed by Philips, East and Pratt in 1981 ([28]). The first hypothesis presented by Lighthill [29] is that the wake of the birds is built up of closed rings shed during the downstroke. The evidence that the downstroke is the only active part of the flapping cycle is represented by the difference of the mass of the muscles that power the two halves of the cycle: the downstroke is powered by muscles representing 15% of the body weight, whereas the upstroke is powered by muscles representing only 1.5% of the total weight of the animal. The "starting vortex" is assumed to be shed at the beginning of the downstroke and is parallel to the trailing edge. Once the lifting regime is established, tip vortices are shed. At the end of the downstroke, the "stopping vortex" is released and the closure of the ring is achieved. The idea of closed vortex rings seemed to be confirmed experimentally, as photographs were taken of birds flying in a dust cloud. The concept of the wake as a chain of vortex rings needed further investigation when a deficiency in

lift was observed from computations in which the total forces was assumed to be produced in the downstroke. A second study by Spedding [27] demonstrated that the upstroke is significantly loaded in forward flight. The difference in the mass of the muscles is realistically due to the most severe flight conditions, like hovering, where the downstroke can be thought of as the phase in which lift is produced. The assumption made is that the wake of the birds is continuous during the stroke. In forward flight it is built up of trailing vortices of constant strength and opposite sign, with the distance between them varying from a wide wake to a narrow wake configuration. This type of wake is called *Concertina Wake* and it seems more in agreement with experimental observations. The two assumptions of the wake formation are illustrated in figures 2.4 and 2.5 for completeness.

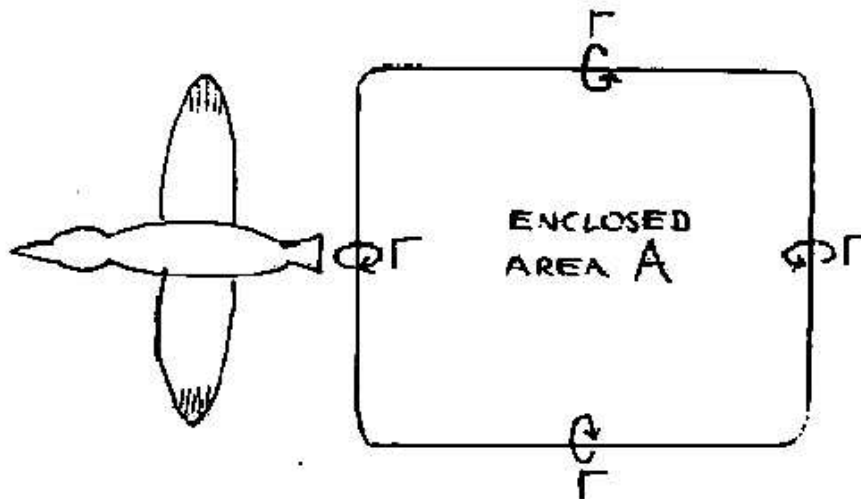


Figure 2.4: Picture of Vortex Ring Wake taken from Lighthill [29]

The theory presented by Lighthill comes from observations and theories developed during the previous decade by biologists and zoologists that attempted to explain with simple mathematical models the flight

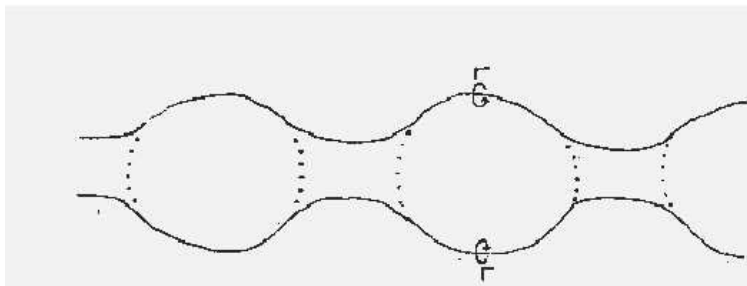


Figure 2.5: Picture of the Concertina Wake, from Lighthill [29]

of animal species. The attention had been focused not on the modelling of the aerodynamics, but on the computations of the power required, the average lift needed to balance the weight, and the efficiency of the flapping wing flight. J. Rayner [25, 26] compares the actuator disk theory and the ring vortex theory. The observations led to believe that a hovering animal's wing beat generates a vortex ring that can be modelled as a circular ring. The momentum jet theory estimated reasonably well the induced power for the flapping motion, but it did not describe the wake formation accurately. Furthermore, the analysis of the forward flight led to the conclusion that the steady aerodynamics does not allow to obtain the high lift coefficients obtained by bats and birds. The theory developed in [25, 26] is less accurate in hovering than in forward flight, because no wake body interaction is taken into account.

2.2.1 Aerodynamics: Understanding of the flowfield

The main characteristic of the flowfield around a flapping wing is the **unsteadiness**, triggered by the periodic motion of the wing. As

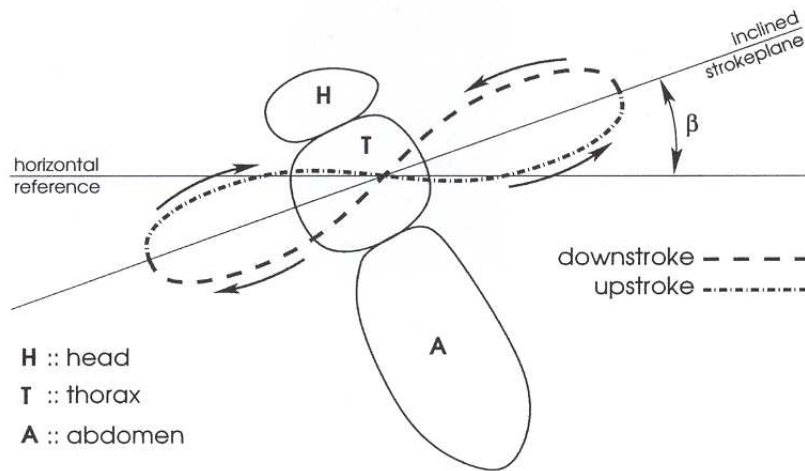


Figure 2.6: Wing tip path of an insect tracing a figure-of-eight profile

Ansari describes in his PhD thesis [30], the tip of an insect-like wing traces a figure-of-eight profile during the stroke. Figure 2.2.1 shows the plane of the motion and the different phases of flapping, upstroke and downstroke. This kind of motion can be decomposed in three fundamental components: vertical displacement, also known as *plunging* or pure flapping, horizontal displacement, called *sweeping*, and rotation around the wing axis, known as *pitching*. The wing sweeps and plunges during the central phase of the half-strokes and it mainly pitches at the end of the run. The sudden rotations at the end (or start) of the strokes give rise to a vortex originating on the wing surface, next to the leading edge, that does not separate immediately, but stays attached to the wing during the mid-part of the strokes. The presence of this vortex was not known until 10 years ago, when the experimental work carried out by Ellington [31] revealed its existence. The main effect of the vortex on the top surface of the wing is the

increase of the lift, that can be explained with the outward induced velocity of the vortex core.

The large frequency at which insects oscillate their wings causes the wake shed at previous strokes to hit the wing surface. This is called *Wake Interference* and is known to have a beneficial effect on the forces produced. It happens mainly in hovering, when there is no freestream speed dragging the wake away, and also in a forward flight condition when a sweeping motion is present. Studies on the effect of the wake hitting the wake surfaces have been conducted for a long time: Schmidt ([32]) in 1965 demonstrated that part of the energy lost in the wake shed from a flapping airfoil can be recovered by placing a steady airfoil behind it. Ansari [30] and Zbikowski ([33]) also show this effect to be important in their studies of flat plates in hovering.

The third characteristic considered in flapping wing MAV applications is the effect of *low Reynolds numbers*. If a speed of about $5 - 10\text{m/s}$ is considered, with a chord of up to 5cm , the Reynolds number is of the order of 10^4 , which makes viscosity not neglectable. The flow is still laminar, therefore no turbulence modelling is needed, but the viscous forces may play a relevant role in the computation of the total forces produced by the wing. Furthermore, as the aim of the flapping motion is to produce thrust besides lift, a viscous drag estimate is necessary to have an accurate prediction of the net thrust.

The objective of the dissertation is to develop a method that is able, as much as possible, to capture all the features of the flowfield that

have been discussed in this paragraph. Therefore the literature review will consist of a presentation of the methods available for the analysis and the consideration of their advantages and drawbacks.

2.2.2 Potential Flow Based Aerodynamics

In 1934 T. Theodorsen [34] published a paper in which the forces acting on a flat plate oscillating in pitch and plunge are expressed analytically as functions of the frequency and the amplitude of oscillation. This was the first attempt to compute the forces acting on a flapping wing, and the theory that actually led to these results is as important as the theory presented by Glauert for steady aerodynamics ([35]).

The importance of the theory lies in the fact that the instantaneous values of the lift and the moment are given as functions of the vertical displacement, the angle of attack and their derivatives up to the second order. In particular, the generic aerodynamic force is thought to be built up of two contributions: non-circulatory, arising from the displacement of the unit mass of air around the wing, and circulatory, due to the generation of the wake behind the wing's trailing edge. The assumptions made by Theodorsen allow to neglect the displacement of the wake from the undisturbed position: the vortices released from the trailing edge are convected with the freestream velocity. The forces predicted with this model are linear in the amplitude of oscillation. The dependence upon the frequency is represented by the Theodorsen function $C(k)$, where k is the non-dimensional frequency.

The aim of the theory devised by Theodorsen is to compute the aeroelastic stability of a very thin airfoil subject to small oscillations around its equilibrium configuration. In the linear case, studying the perturbation around a given configuration is equivalent to studying the perturbation around the undisturbed configuration. In actual facts, the results obtained by Theodorsen were confirmed by experimental investigations on symmetric wings of very high aspect ratios excited with sinusoidal motions in pitch and plunge. Theodorsen theory still stands today as one of the fundamental theories in the field of potential flow unsteady aerodynamics. It is a useful tool for a quick computation of the forces acting on an airfoil and it can be used as a benchmark to test numerical methods based on potential flow analysis.

A few years later I.E. Garrick [36, 37] made further developments to the model of Theodorsen and succeeded in finding an analytic expression for the thrust produced by a flapping flat plate. The arguments used to get to such expression are based on the energy conservation in the wake and the existence of a suction force at the leading edge of the airfoil. The assumption of inviscid flow and small amplitudes are still present in this model, therefore the lift is the vertical force perpendicular both to the flow direction and to the chord, whereas the thrust is the force parallel to the chord. Furthermore, the thrust does not take into account any kind of drag, because the assumptions of the theory do not allow to predict it. Nevertheless, the model of Garrick is able to predict fairly well the trend of the thrust for a flat plate undergoing sinusoidal oscillations. In pure plunge, the thrust produced by

the airfoil is positive for all the frequencies of oscillation, as shown in figure 2.2.2 and proved in [38, 39]. Moreover, if the thrust coefficient is defined as the ratio between the thrust and the dynamic pressure, it is found to be proportional to the square of the reduced frequency, as it will be confirmed in chapter 4. If the reduced frequency is defined as

$$k = \frac{\omega b}{U_\infty}$$

where ω is the angular frequency, b is the semichord and U_∞ is the freestream speed, then the average thrust coefficient is found to be

$$C_T = \pi(kh)^2 (F^2 + G^2)$$

with h being the non-dimensional amplitude of oscillation, F and G the real and imaginary parts of Theodorsen function, and b the length used for adimensionalization. Another important result that can be obtained from this model is that the frequency of the thrust for a plunging airfoil is twice the frequency of the lift. This will again be confirmed in the computations.

As shown in figure 2.2.2, for a plunging airfoil the composition of the forward speed and the vertical speed causes the thrust to be tilted forward. For a pitching airfoil, Garrick shows that the average thrust is negative up to a reduced frequency of about 2, being positive for all the other frequencies. A second important parameter in the study of flapping wing performance is the efficiency, defined as the ratio

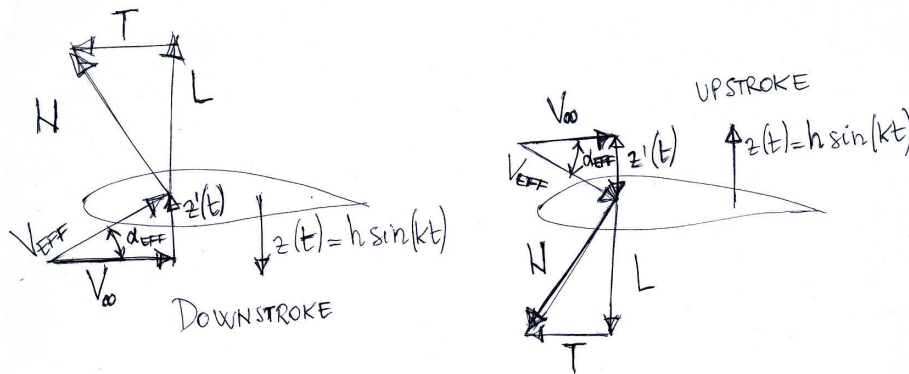


Figure 2.7: Sketch of the total force produced by a plunging airfoil

between the power output and the power input:

$$\eta = \frac{TU_{\infty}}{P_{in}} = \frac{C_T}{C_P}$$

For a plunging airfoil, the efficiency is shown to decrease when the frequency increases, being equal to 1 for $k = 0$ and approaching asymptotically a value of 0.5 for large frequencies. Moreover, the theoretical efficiency does not depend upon the amplitude of oscillation. This allows us to conclude that the theoretical optimum combination to maximise both thrust and efficiency is flapping at relatively low frequencies and large amplitudes.

Although much work has been done on flapping wing flight from Theodorsen and Garrick onwards, most of the methods developed by researchers are still based on the same assumptions that they made. The main limitation of the models just described is the small oscillation assumption, that does not allow to obtain a correct representation of the flowfield when amplitudes become larger. The main consequence

of this assumption is that the wake can be treated as a flat surface lying behind the airfoil. This is not the case for insect-like wings, in which the deformation of the wake is exploited to obtain higher values of the forces. Therefore the small oscillation assumption needs to be removed to have a better representation of what the wake looks like once it has been released from the trailing edge of the airfoil.

Fairgrieve and De Laurier [40, 41] showed how to improve the model described by Theodorsen and Garrick in order to account for large oscillations. A time-domain approach is considered, where one vortex is released at each time step from the trailing edge. The main assumption is the validity of Kelvin's law, that states the conservation of vorticity for all the time steps. Therefore, if a vortex of strength Γ is released at time $t_j = j\Delta t, j = 1, \dots, N$, then a circulation bound to the airfoil builds up such that the algebraic sum of the present vortex, the vortices released at previous time steps and the bound circulation is zero. The circulation bound to the airfoil is built up of two contributions: a quasi-steady part, caused by the motion of the airfoil in the air surrounding it, and an unsteady part, coming from the wake convected in the field. This decomposition is similar to the description found in Theodorsen, with the difference that in this case there is no need to compute the values of a transcendental function. After the circulation on the airfoil has been found, Kelvin's law allows to compute the amount of circulation that needs to be shed in order to have the total circulation in the field equal to zero.

Three models to describe the wake are presented in [40] and [42]:

1. **Planar Wake:** it is the same model presented in Theodorsen, where the wake is released and convected with the freestream speed;
2. **Frozen Wavy Wake:** the wake is released at each step in the position occupied by the airfoil's trailing edge, and remains frozen in that position. The result of this model is a wake that describes the pattern of the trailing edge;
3. **Deformable Wavy Wake:** once the generic element has been released in the position occupied by the trailing edge, convection occurs based on the interaction of potential flow vortex elements.

The second model is thought as a correction to the flat wake model, yielding more accurate results. The novelty of the out-of-plane-wake model is that the thrust is not obtained on the basis of a suction force; it can be obtained from the vertical displacement of the wake elements. A pair of vortices in different positions generate a force whose absolute value is proportional to their strength times their reciprocal distance and whose direction is perpendicular to the line connecting their positions. On the basis of this theory, the flat wake model can only generate lift. For a model in which the wake is not confined to a plane behind the airfoil, the total force can be decomposed in thrust and lift, respectively parallel and perpendicular to the flow direction. The whole theory is based on the assumption that the momentum of the

overall system can be obtained as the sum of the momentum of the vortex pairs in the flowfield. Decomposing the vorticity bound to the airfoil in the same number of vortices shed in the flowfield, all concentrated at the same location, then N -steps vortex pairs will be present in the field.

The analysis carried out is limited to flat plates, but the deformation of the wake can be taken into account and compared with the results obtained with the flat wake. One point that must be kept in mind is that the amplitude of the oscillations is not required to be small, as long as it is remembered that the model is still inviscid and no separation is predicted. This point is important in all the inviscid methods that are based on the potential flow assumption. The aerodynamics of an oscillating airfoil can be computed at arbitrary pitch and/or plunge amplitudes, nevertheless at that amplitudes the flow over the airfoil may have separated already. The potential flow solver simply cannot predict it.

2.2.3 Panel Methods

The method presented above is semi-analytical, because it uses a discrete time step to advance the calculation. The main limitation is that general wing configurations cannot be analyzed. To allow geometries of any kind to be computed, a fully numerical method is required. **Panel Methods** are still based on potential flow assumption, with the difference that airfoils or wings can be taken into account. The ad-

vantage of panel methods compared to CFD solvers is the run time required for the analysis: CFD may take hours or even days for unsteady computations of flapping wings, whereas panel methods take minutes or even seconds. Therefore they are widespread in the computational fluid dynamics community, especially in the preliminary design phase of the aircraft, when a massive number of computations is needed to decide what the best configuration is. This section is intended to give an overview of the ways in which panel methods are used for computations of arbitrary wing geometries. Technical details on the theory underlying panel methods are given in chapter 3. For a more rigorous derivation of the panel methods from potential flow assumptions the reader is referred to [43].

The success of panel methods is mostly due to their use in computational aeroelasticity. Strictly speaking, they are not considered in the family of the methods used for computational fluid dynamics, even though they can predict qualitatively and, in some cases, quantitatively, the main features of the flow field under the assumption of irrotational flow. Thanks to the development of computational methods in aeroelasticity and the necessity to run calculations in which the structure and the flow are coupled together, the use of panel methods has become increasingly frequent in the last couple of decades. A panel method is used in NASTRAN for the coupling of structure and aerodynamics for the prediction of aeroelastic stability and control. The module that embodies this feature is **Flight Loads** and is based on potential flow for the computation of the aerodynamic forces acting

on the structure. The version of the method used in NASTRAN neglects the thickness of the body, that is represented by a zero-thickness surface, on which the forces and the displacements are computed and subsequently interpolated onto the structural grid.

Moreover, in the field of Flapping Wing Aerodynamics, the difficulty to generate the grid at each time step has left room for the developments of alternative methods to the traditional CFD solvers. Panel methods are the most widespread, for their ability to predict reasonably well the forces as long as the flow is attached to the surface. Jones and Platzer ([3, 4, 5, 6, 7, 8, 9, 10, 11]) developed a tool for the analysis of the aerodynamics of Flapping Wings configurations and validated it against experimental results obtained in the wind tunnel and with flying models. The panel method of Jones and Platzer is based on the work of Basu and Hancock [44], that is an extension to unsteady motion of the work of Hess and Smith [45]. The equation of Laplace is obtained introducing the definition of potential in the mass conservation equation. It is not dependent upon time, and it needs conditions on all the boundaries. If $\mathbf{x} = (x, y, z)$, then the system of equations can be written as:

$$\left\{ \begin{array}{l} \nabla^2 \phi = 0 \\ \frac{\partial \phi}{\partial n} = v_b \cdot n, \mathbf{x} \in \partial B \\ \frac{\partial \phi}{\partial n} = 0, \mathbf{x} \longrightarrow \infty \end{array} \right. \quad (2.1)$$

In the above equation ϕ represents the perturbation potential. The

total potential will be represented by the symbol Φ and is the sum of the free-stream velocity potential and the perturbation potential as follows: $\Phi = \Phi_\infty + \phi$. Unsteadiness is fed indirectly into the system through the impermeability condition, that says that at each time step the velocity of the flow over the body must be the same as the velocity of the body. Therefore, if the airfoil is performing sinusoidal oscillations in pitch and/or plunge, the bound circulation changes and the circulation shed in the wake is different at each time step. As shown in the system 2.1, an important advantage of potential-flow-based methods over CFD is that a solution is sought in a form such that the flow at infinity satisfies the condition of freestream velocity. Basu and Hancock [44] and more recently Jones and Platzer [3, 4, 5, 6, 7, 8, 9, 10, 11] used vortices to represent the airfoil's surface and the wake shed. At each time step, the no-through flow boundary condition is fulfilled on the midpoints of the panels and one wake element per time step is shed. The strength of this element is determined by the condition that the circulation in the field remains the same. The position of the wake elements is updated at the end of each time step, based upon the influence of all the wake elements and the vortices of the body. One more feature that is embedded in the panel method is the computation of both the strength and the orientation of the last element released. It has already been explained the way in which the strength is found. The orientation of the panel is important as well, as this results in a different position at which the vortex is shed and in different influence of the vortex on the body panels. Several

authors fix the orientation of the wake panel attached to the airfoil, as they consider it a minor issue in the development of the panel method. Katz and Plotkin [12] suggest that the wake element leaves the trailing edge at an angle equal to the average of the angle between the upper and lower trailing edge panels. Although this might seem reasonable, Jones and Platzer claim that their method yields more accurate results.

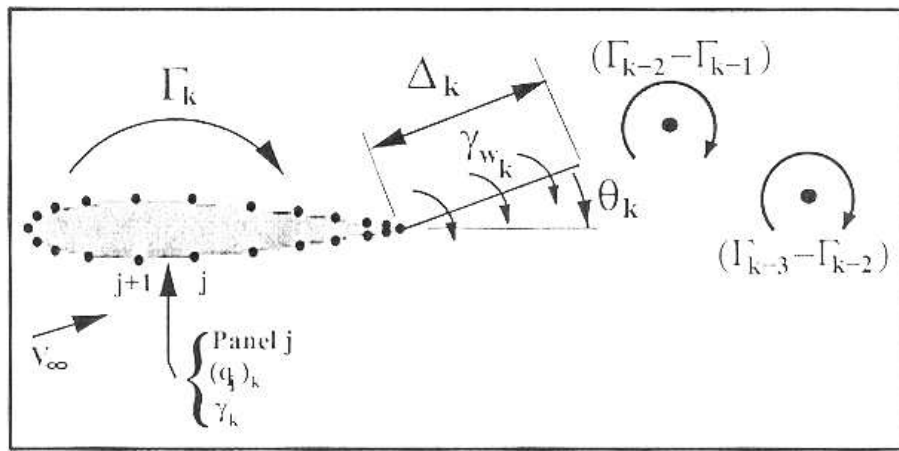


Figure 2.8: Sketch of the wake release process, taken from Jones [4]

Figure 2.2.3 shows the principles of the method employed by Jones and Platzer for the wake release algorithm: the orientation of the panel attached to the trailing edge is equal to the orientation of the velocity with which the flow leaves the trailing edge. The velocity is not known *a-priori*, therefore an iterative procedure is needed to compute its value. The method used is a Newton-Raphson scheme, where an initial orientation is prescribed. Convergence is achieved when the orientation of the velocity vector at two consecutive time steps is below a given tolerance. Jones and Platzer claim the orientation to be fun-

damental to describe the non-linearities occurring in the flow at high frequency of oscillation. The next chapter will explain how the panel method developed in the present investigation will deal with the panel orientation and what the differences with the Jones-Platzer method are.

The core of panel methods is the computation of the singularities placed on the body surface. The basic solutions of Laplace equations in two dimensions are basically of three kinds:

1. **Source/Sink:** the velocity component of the flow is radial, the potential is $\Phi = \frac{\sigma}{2\pi} \log r$, where σ is the strength and r is the distance between the source and the point at which influence is computed;
2. **Doublet:** it is obtained letting a sink and a source approach each other, maintaining constant the product of the strength and the distance. The potential in this case is $\Phi = -\frac{\mu \cdot \mathbf{r}}{2\pi r^2}$, with μ equal to the strength of the doublet;
3. **Vortex:** the velocity induced by a point vortex is tangential and is given by the Biot-Savart law. If Γ is the vortex strength, the potential is given by $\Gamma = -\frac{\Gamma}{2\pi} \theta$ and the induced tangential velocity is $q_\theta = \frac{\Gamma}{2\pi r}$.

A detailed description of the differences arising in the pressure coefficient computation when using different kind of singularities may be found in [12]. Although point-concentrated singularities are allowed,

the development of accurate panel methods requires the use of higher order solutions. A constant strength singularity is a solution whose strength is constant over the panel length. For the vortex solution, the strength is equal to the circulation density over the panel length. If the total circulation of the panel is needed, the density has to be multiplied by the panel length. The demand for higher accuracy has led to the use of higher-order distributions of singularities: other than point concentrated and constant density, it is possible to employ linear and quadratic singularities. This means a greater number of equations to be solved, as additional conditions need to be satisfied to have a smooth match between two panels. As an example of constant strength solution, figure 2.2.3 shows doublets of constant strength. As [12] point out, a constant strength doublet is equivalent to a vortex pair concentrated at its vertices.

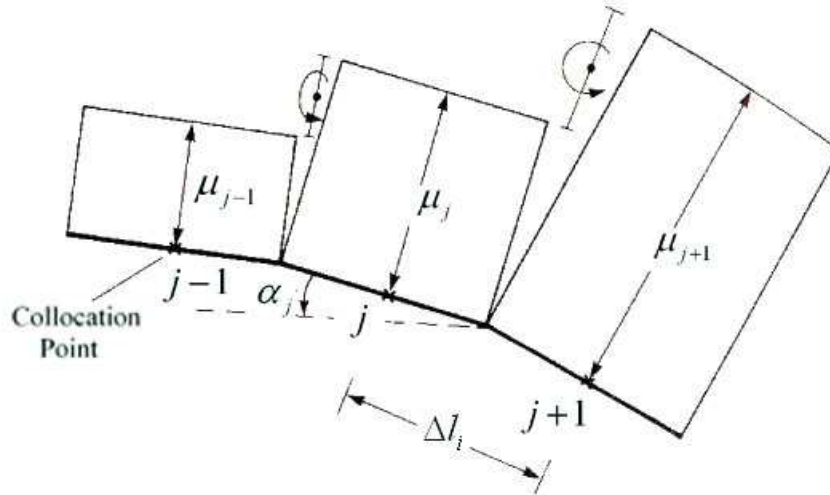


Figure 2.9: Constant Strength Doublets, taken from [12]

Inviscid flow solutions fulfill the condition that the flow direction is

tangent to the panels. As the discretization of the body is carried out with straight panels, the condition of tangency is imposed in one point, defined as the *collocation point*. The velocity at the collocation point is a linear function of the strengths of the singularities in all the flow field, hence the tangency condition leads to solving a N-by-N linear system in which the strength of the singularities in correspondence of all the collocation points are unknowns. Another central issue in the solution of the system is the constraint that the vorticity remains constant and that the velocity at the trailing edge remains bounded. There has been a lot of discussions on the way in which the Kutta-Joukowski condition has to be applied. It has already been shown the way Jones and Platzer fulfill it [4]. Maskell [47] remarks that ” *On the assumption that the velocity field is everywhere finite and continuous-apart from permissible tangential discontinuities across free vortex sheets- it is concluded that vorticity can be shed from a sharp edge, in a two-dimensional unsteady flow of vanishingly small viscosity, only along a vortex sheet that is tangential to the surface of the body at the edge. The relevant tangential direction-where two are possible- is determined, unambiguously, by the sign of the shed vorticity*”. This statement is based on the assumption that the flow cannot separate upstream of the trailing edge and the vortex sheet originates from an interaction of the outer and the inner flows, where the inner flow consists of a laminar boundary layer. The method of matched asymptotic expansions, as shown in [48], justifies the presence of a vortex sheet in correspondence of the trailing edge of an airfoil. Davi and

al. [49] suggest the use of an iterative procedure similar to the one presented by Jones and Platzer, with the difference that equality of the pressure between lower and upper panels is imposed. In the unsteady case the computation of the pressure coefficient requires the knowledge of the potential and the velocity values over the trailing edge panels, not known *a priori*. Therefore an iterative Newton-Raphson scheme has been used to find an equality of the pressures.

The use of panel methods is advantageous when run time is a concern. The limit of the method remains the impossibility to describe separation upstream of the trailing edge. This may be important when dealing with flapping wings, therefore alternative methods have been investigated in order to bridge the gaps left by the panel method. These will be described in the following sections.

2.2.4 Vortex Methods

Vortex methods are based on the solution of the Navier-Stokes equation expressed in terms of the vorticity. A complete derivation of the theory from general principles can be found in [50, 51, 52]. For a two dimensional incompressible flow the Navier Stokes equation takes the form:

$$\frac{D\omega}{Dt} = \frac{\partial\omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \nu \nabla^2 \omega \quad (2.2)$$

The time derivative on the left hand side of equation 2.2 indicates

that the particles carrying vorticity are material elements and can be tracked in a Lagrangian way. Due to the small region in which the vorticity is confined, vortex methods are advantageous compared to CFD methods because of the smaller computational effort required. On one side, they can be considered as Lagrangian solvers, because no discretization is needed and the particles carrying vorticity are followed during the convection and diffusion processes. On the other sides, grids can be used to regularize the particle locations after a fixed number of time steps. The main characteristic of the vortex methods is the attempt to find a solution to equation 2.2 in two sub-steps: convection and diffusion. The formulation of the two steps is given in the following equations:

$$\begin{cases} \frac{d\mathbf{x}_i}{dt} = \mathbf{u}(\mathbf{x}_i, t) \\ \frac{d\omega}{dt} = \nu \nabla^2 \omega + B.C. \end{cases} \quad (2.3)$$

1. **Convection:** The particle position is updated based on the Biot-Savart law, where all vortices present in the flowfield are considered. The completion of this step may become computationally expensive if more than a thousand particles need to be computed. Techniques to reduce this time to an acceptable level have been developed;
2. **Diffusion:** This step needs to be carried out in order to enforce the no-slip boundary condition on the body surface. At the end of the convection step, a slip velocity is present on the body surface.

This can be cancelled if a vortex sheet of appropriate strength is built up on the surface of the body. This vortex sheet is subsequently released into the flow and diffused to other particles via different techniques.

Since the time when Chorin [53] presented his vortex method in 1973, the effort of researchers and scientists around the world has concentrated on the improvement of the convection and diffusion processes in order to yield more accurate results and to approach the reliability of CFD methods. Often considered as *back-of-the-envelope* answers, today's vortex methods can be thought of as an alternative to CFD when a quick answer is sought for a problem. The assumption behind a vortex method is that the velocity field can be decomposed in its potential and rotational components. The potential part is found satisfying the condition of no-through-flow when solid walls are present. On the other hand, the rotational velocity is found by means of Biot-Savart relation. Lewis [46] shows how a panel method can be used in order to find the strengths of the vortices on the body surface. These vortices are subsequently shed in the field and subject to the convection-diffusion process. Ansari [30] presents a model that is based on the shedding of two vortices from trailing and leading edges of a flat plate oscillating in pitch and plunge. No diffusion is considered in his model, but the vortex strength is *spread* over a circular area around the vortex core. This technique is used in order to avoid infinite induced velocities when two vortices come too close to

each other. The model of Ansari had already been presented in Lewis [46], although the conditions used to determine the vortex strengths are completely new. M.A. Jones [54] presents a model very similar to Ansari, still valid for a flat plate performing large amplitude oscillations in pitch and plunge. The restriction to this model is that the airfoil must exhibit high angle of attack, because of the instability that the numerical algorithm develops when vortex particles drift too close to the plate surface. It is fairly accurate to shed vortices from fixed points when the geometry of the body allows to assume that separation occurs from those points. For instance, in the case of a wedge-shaped airfoil, it can be assumed that the flow separates from the corners delimiting the vertical edge, as shown in figure 4.52. In the case of bluff bodies this assumption does not hold in the same way, because the separation point may move back and forth under the interaction of the boundary layer and the external flow. Figure 2.11 shows how a simple result can be obtained in the case of a circular cylinder, for which the separation points are not fixed.

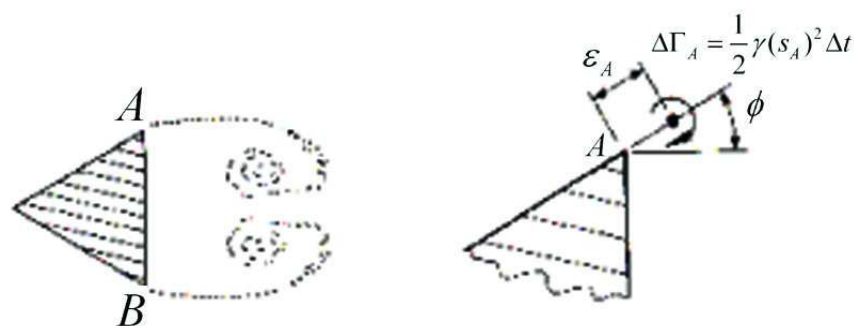


Figure 2.10: Modelling of Separation from Wedge Shaped Body, from [46]

For more complicated configurations, as an airfoil or a wing can be,

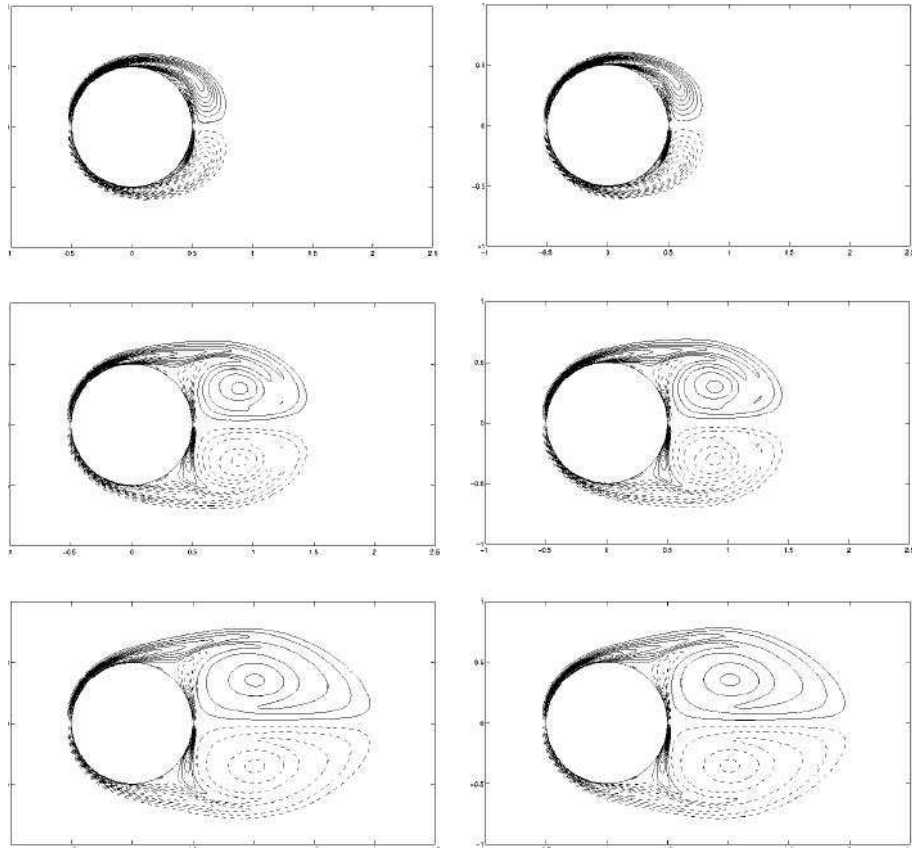


Figure 2.11: Separation from a cylinder predicted using a Vortex Particle Method ([50]). From top left to bottom right, the separation is plotted for different time steps

the assumption of separation at fixed points may not be strictly valid, therefore a model that takes into account the effect of the motion and the influence of the wake on the separation needs to be considered. Discrete Vortex Methods (DVM) originated from the representation of the vorticity field with blobs of fixed or variable size, whose influence decays with the distance from the core, and with a finite value of the induced velocity at the centre of the blob. The shape functions used for the approximation need to be defined in order to guarantee the convergence of the discrete field to the continuous field of the vorticity. A rigorous convergence analysis is performed and in [50], showing that as the size of the blobs tends to zero, the blobs approximate the point vortices and the continuity of the field is regained. The approximation of point vortices with finite size blobs is indicated as *mollification* and it is given in the following formula:

$$\omega_{\epsilon}^h(\mathbf{x}) = \sum_p \alpha_p \zeta_{\epsilon}(\mathbf{x} - \mathbf{x}_p^h) \quad (2.4)$$

where α_p is the strength of the vortex particle and ζ_{ϵ} tends to a dirac distribution as the core ϵ tends to zero. So far, the *cut-off function* ζ_{ϵ} is completely arbitrary, the only restriction being that it has to approximate the dirac function when the core size decreases. For the approximation to be consistent, Cottet and Koumoutsakos show that the moments of order higher than one need to be zero, as for the dirac distribution. It is said that a cut-off is of order r if:

$$\begin{aligned}
\int \zeta(\mathbf{x}) d\mathbf{x} &= 1 \\
\int \mathbf{x}^i \zeta(\mathbf{x}) d\mathbf{x} &= 0, \text{ if } |\mathbf{i}| < r - 1 \\
\int |\mathbf{x}|^r |\zeta(\mathbf{x})| d\mathbf{x} &< \infty
\end{aligned} \tag{2.5}$$

In his book ([46], pag.400), Lewis accounts for a growth of the vortex core with time, the growth-law being very similar to the gaussian blob function. The use of *blobs* to represent the vorticity field has not become widespread until the work of Cottet and Koumoutsakos [50, 55, 56], and Leonard [57, 58]. Eldredge ([59]) outlines that it is fundamental, in the approximation, for the radius of the vortex core to be large enough to allow the vortices to overlap each other. After having obtained a discretization of the field through introduction of vortices, the velocity can be obtained via Biot-Savart law, that reads as:

$$\mathbf{u}^h = \mathbf{K}_\epsilon \star \omega^h \tag{2.6}$$

where the $\star$ operator indicates convolution. The accuracy of vortex method depends mostly on the convection algorithm used to advance the position of the vortices. Krasny [60] presents results on the instability of a vortex sheet with an initial sinusoidal shape convecting under its self-influence. He makes use of a de-singularization kernel slightly different than the ones used by Cottet and Koumoutsakos. The stability study performed in his paper shows that no an-

alytic solution exists for such a vortex distribution after a fixed time ($t = 0.375$), due to the spiralling of the sheet. The de-singularization procedure allows to obtain a detailed description of the sheet shape long after the critical time step. Convergence is studied varying the de-singularization parameter ϵ and the number of points representing the sheet. The time integration scheme to advance computation is the fourth order Runge Kutta, therefore high accuracy is guaranteed.

When the flow domain is bounded, for example due to the presence of a body in it, the blobs next to the body need to be a-symmetrical, in order not to have circulation inside the body contour. Particles drifted too close to the solid surface need to be snuffed out or removed from the flowfield (see [46]) and their circulation still taken into account in the solution of the system to find the vortex sheet strength. In this way circulation conservation is guaranteed. As the flow charts in figure 2.2.4 show, the solution at each time step requires the use of a panel method in order to find the strength of the singularities over the body. The left and right flow charts are different because they perform the convection step in two different ways: in the left one the position and velocity of the vortices are affected by the singularities on the body, whereas in the right one they are not taken into account. The time integration scheme considered in the example described by the flow charts is a higher-order Euler method, therefore one step requires several sub-steps to be executed before advancing to the next step. In the right flow charts, the panel method is applied just at the first sub-step, and the singularities on the body are not modified anymore

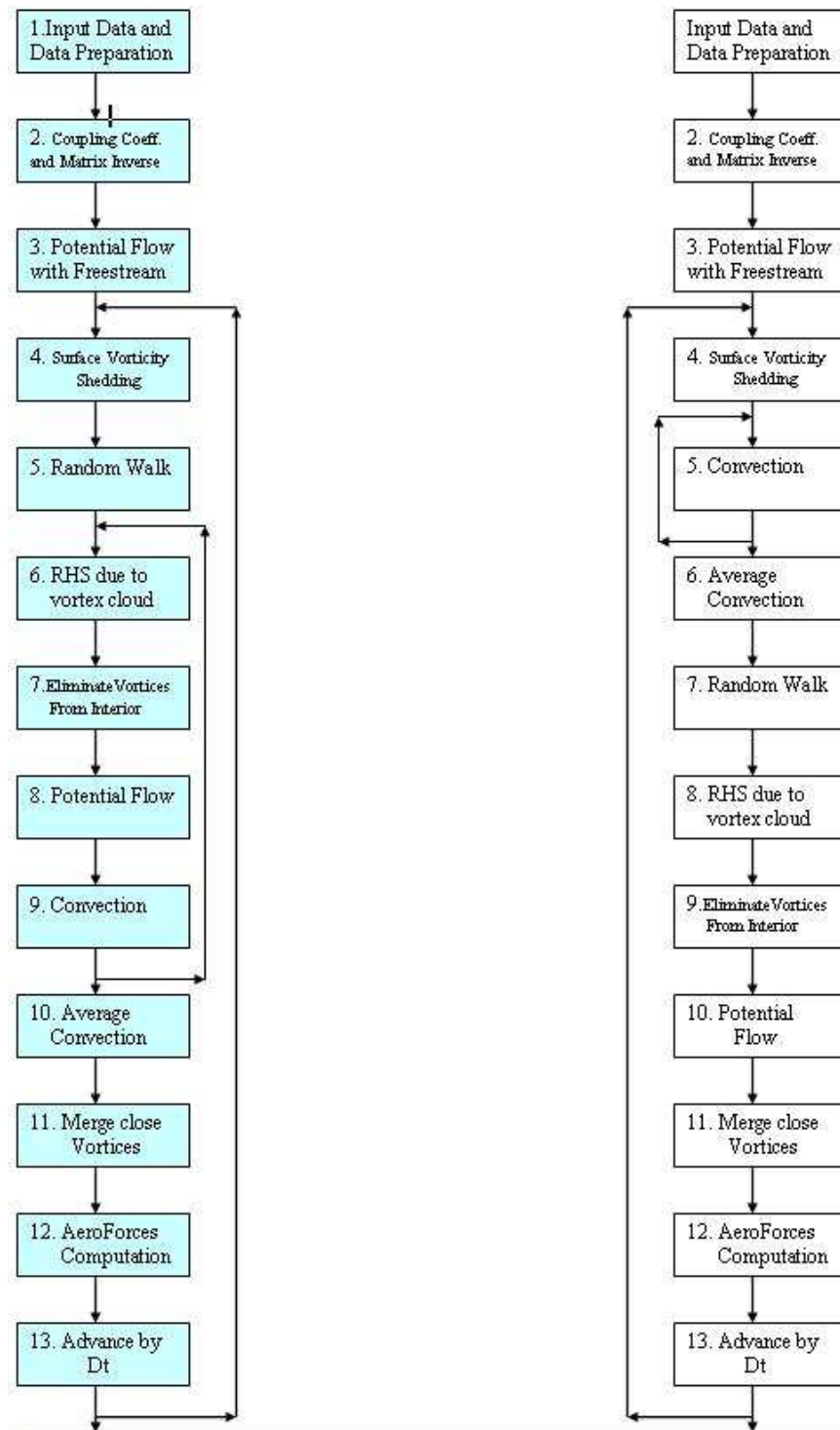


Figure 2.12: Flow Chart indicating the solution steps, as in Lewis [46]

for all the other sub-steps. This is clearly an approximation, because the strength of the vorticity on a panel depends on the position of all the vortices in the field. If the distribution of the vortices changes, the strength of the singularities need to be recomputed to satisfy the no-through-flow boundary condition.

The second part of equation 2.3 describes the diffusion of vorticity among particles. This process is based on the solution of the heat equation, with the vorticity replacing the temperature field. Chorin ([53]) proposed the Random Walk Method (**RWM**) to describe the diffusion of vorticity. This consists of distributing vortex particles in the xy plane in order to approximate the initial vorticity distribution and then moving them according to the law:

$$\begin{aligned}x_i^{n+1} &= x_i^n + \eta_1 \\ y_i^{n+1} &= y_i^n + \eta_2\end{aligned}$$

where η_1, η_2 are Gaussianly distributed random variables. After N steps the distribution will approximate, in a statistical sense, the solution. This method is based on the analytic solution of the heat equation, that can be expressed as in [61]:

$$\omega(\mathbf{x}, t) = \int_D G(\mathbf{x} - \mathbf{x}_0, t) \omega_0(\mathbf{x}_0, \tilde{t}) dD_0 \quad (2.7)$$

with ω_0 being the initial vorticity distribution and $G(\mathbf{r}, t)$ the Green's function to the diffusion equation

$$G(\mathbf{r}, t) = \frac{1}{4\pi\nu t} \exp\left(-\frac{|\mathbf{r}|^2}{4\nu t}\right) \quad (2.8)$$

Deterministic methods differ from stochastic one because they are based more on the Eulerian diffusion model than the Lagrangian one used in the Random Walk. The method that has received the largest attention is the Particle Strength Exchange (**PSE**), based on the redistribution of strength among particles. The theory underlying the method is given in [62] and [63]. Recently it has been used by Eldredge ([64, 65, 66, 67, 68, 69, 70, 71]) and Barba ([72, 73]) for the solution of the vorticity field in bounded and unbounded domains. In the present method, the diffusive action described by the second row of equation 2.3 is taken into account via a modification of the particle strengths. The Laplacian operator is replaced with a symmetrical integral operator, more suited to the use of particles. As for the discretization of the flowfield through the introduction of blob functions, the Laplacian can be approximated to any order of accuracy if the kernel function is properly chosen. Since the Laplacian differential operator is replaced with an integral operator, defined as:

$$\nabla^2 \omega(\mathbf{x}) = \frac{2}{\sigma^2} \int \eta_\sigma(\mathbf{x} - \mathbf{y}) (\omega(\mathbf{y}) - \omega(\mathbf{x})) d\mathbf{y} \quad (2.9)$$

and the integral operator is discretized over the particle positions, this technique is prone to be treated with a regular spacing among particles. The performance of the PSE is maximised when the particles overlap. Therefore a grid is used for carrying out the diffusion process

and the particles strength is interpolated over the grid centroids. The re-initialization of the vorticity over the grid points needs to be carried out every fixed amount of time steps, to ensure that the particles overlapping is re-established. In practice, each point on the grid is affected by neighbouring particles only, due to the decay of the influence when the distance between particles increases. Ploumhans and Winckelmans ([74]) say that only particles within 5σ distance need to be taken into account for interpolation. Another issue in the PSE is whether fixed core or variable core particles should be used. The advantage of using variable core particles is that the elements in the far wake can be treated with a coarser resolution, thus increasing the efficiency of the method. Ploumhans and Winckelmans ([74]) give proof of convergence of the PSE for this latter case. Other methods are available for the computation of the diffusion, and they are described in the book of Cottet and Koumoutsakos [50], therefore they will not be described in this survey.

A central issue in the vortex particle method technique is the efficient computation of the convection step: particles number might become as high as $N = 10^5$ in the flowfield, therefore the amount of operations to perform would be $O(N^2)$, not affordable for a vortex method. The Fast Multiple Method (FMM) has been introduced in order to reduce the amount of operations to $O(N \log N)$ or to $O(N)$ in the best hypothesis. FMM is described in [50] and [57]. It was originally developed for three-dimensional N-body simulations and then modified for the use in Vortex Particle Methods. The particles in the flowfield are organized

into a hierarchy of clusters. The computational space is divided using cubes enclosing the particles. The finer the division, the greater the accuracy. In order to minimise the work, the size of the cluster needs to be maximised. The influence of a cluster of particles over a vortex can be computed accurately as long as the distance between the cluster and the vortex is much greater than the size of the cluster itself. The computational structure used to represent the physical domain is a *tree*, where each node is connected to a *parent* node and has *children* nodes. The process is started at the finest level, and then moves up to compute the influence of parents nodes to each other, until the top level is reached. The reduction in the amount of operations is huge, and it allows for the representation of the flowfield with a large number of vortices.

Another central issue in the development of VPMs is the redistribution scheme applied every few time steps in order to regularize the particle positions. The attention in this case has to be paid to the conservation of the vorticity and the moments of greater order. The application of this process is necessary due to the strain introduced in the particle positions when convection and diffusion are carried out for several time steps. The particles positions are used as *quadrature points* for the computation of the Laplacian, therefore a greater accuracy is obtained if the particles are uniformly spaced over the domain. The operation consists of transferring the vorticity values from an irregular grid, the Lagrangian positions of the blobs, to a regular mesh built up around the body.

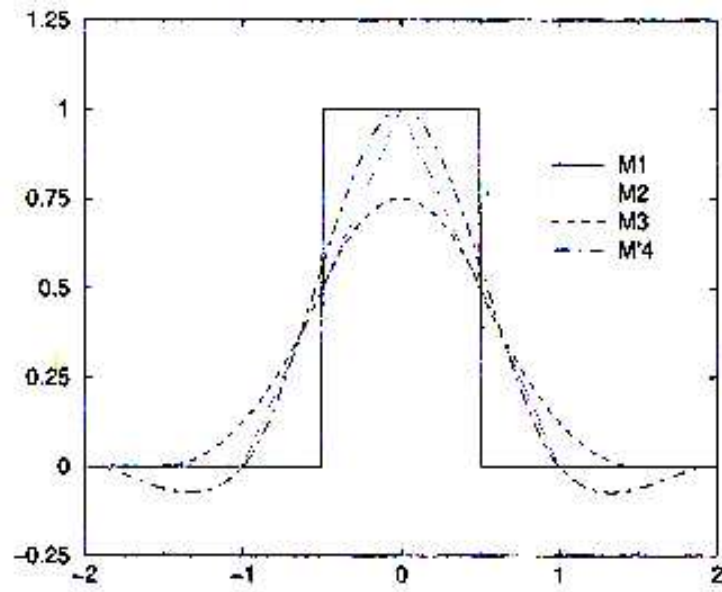


Figure 2.13: Examples of polynomial interpolating functions, from [50]

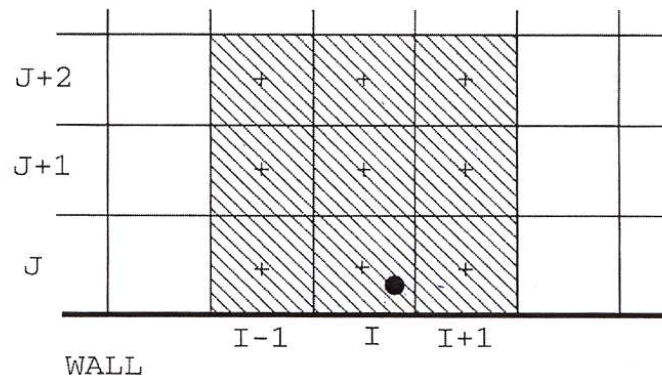


Figure 2.14: Interpolation of vorticity from Lagrangian element (●) to grid points (+) from [50]

This interpolation process needs the definition of smooth and accurate interpolating functions, with some examples shown in fig. 2.13. One way in which these functions can be extended to vortices with a two-dimensional support is by the product of two functions of the same kind, each one dependent on one direction only:

$$\Lambda(x, y) = \Lambda_I(x)\Lambda_J(y)$$

For bounded domains, the interpolating functions need to be modified in order for the circulation not to be diffused into the body contour. therefore an asymmetric function is employed in this case, for example (see [50] and [56, 75]) interpolating first in the tangential direction and then in the normal one. Figure 2.14 shows this situation, where a vortex particle is close to the body and its circulation has to be transferred to grid points in its proximity. A group of grid points to which circulation is transferred is spotted first, then each point is given an amount of circulation inversely proportional to the distance from the giving particle.

2.2.5 CFD Methods

A brief presentation of the work carried out with CFD solvers is given in this section for the sake of completeness. Even though separation is a common phenomenon occurring on flapping wings and the separation region could be resolved accurately if a proper CFD solver

with moving grid is used, the attention of researchers and scientists has most of the time concentrated on finding alternative methods to CFD. Zbikowski [76] uses a Navier Stokes solver in order to predict the formation of a leading edge vortex on a flapping wing and to highlight the benefit of this vortex for insect-like wings. In particular he achieves a solution based on a three dimensional, Runge-Kutta, explicit, code. The dual-mesh, edge-based, finite-volume code is used with hexahedral cells. Zbikowski points out how dynamic remeshing is a source of error and to avoid this at each time step the equations for the volume and surface conservation are solved.

The advantage of the Navier Stokes solvers is the possibility to compute the flow around complex configurations, provided that the mesh is built in an appropriate manner. Togashi and al. [77] present the solution of the flowfield around a hornet in forward flight. The configuration investigated includes antennas, legs and sting, realistically reconstructed in the computational space with the use of unstructured grids. The advantage of the overset unstructured grid is the possibility to treat large movements of wing without any difficulty. The inclusion of wing flexibility is further suggested in order to have a better comparison with experimental results.

The desired and undesired effects of dynamic stall on a pitching airfoil are investigated by Wernert and al. [78] with both experimental means and a Navier Stokes solver at a Reynolds number of 3.73×10^5 . A turbulence model is also embedded in the computation in order to

correctly describe the flow in the separation region. A study on the effect of the Strouhal number and the reduced frequency on pitching airfoils is also carried out by Ramarurti and al. in [79, 80, 81, 82], where a finite element incompressible flow solver with unstructured grids is used for the analysis. The question whether the flow around oscillating airfoils is dependent upon the Strouhal number or the reduced frequency is still unanswered, although many studies have been performed to this purpose. Young [83] in his PhD thesis has developed a CFD solver and compared the results with a potential flow solver in order to investigate the effect of both reduced frequency and Strouhal number on the wake shedding process and the forces developed on the airfoil. Similar investigations have been carried out by Tuncer and al. ([84, 85, 86, 87, 88]), Neace ([89]) and Cebeci ([90]), in which single and multiple flapping airfoils are taken into account. For the single airfoil flapping case, the parameters varied are the amplitude, the phase shift between pitch and plunge motions and the reduced frequencies. For the multiple airfoils case, the a flapping airfoil is placed in the wake of a steady one, and the effect on the downstream airfoil is observed. In particular, the steady airfoil is shaped in order to facilitate separation. A flow reattachment is observed both experimentally and numerically due to the presence of the flapping airfoil. The distance between the two airfoils and the amplitude of flapping are varied in order to assess their contribution to the coupling. A recent research at MIT ([91, 92]) has presented a deep comparison between panel methods and CFD methods.

The discussion about CFD methods used in unsteady aerodynamics for flapping wings will be integrated with the CFD methods used in computational aeroelasticity. As largely explained earlier on, CFD methods will not be employed in the remainder of the thesis. They will be used, where possible, as a means of comparison, in particular when experimental results will not be available.

2.3 Aerodynamics:Experimental Results

Although this dissertation is focused mainly on the numerical analysis for the prediction of the flowfield around a flapping wing, it is reckoned necessary to illustrate up-to-date progress on wind tunnel tests and experimental models of flapping wings. The main source of experimental work carried out on flapping wings has been, for the purpose of this dissertation, the work done by Kevin Jones and Max Platzer at Naval Postgraduate School (US). Besides developing a panel method, performing Navier Stokes computations for 2D and 3D flapping wings and building a flapping wing aircraft prototype of their own, they have provided several experimental results useful for comparisons both with flapping airfoils and wings. In [13] results are presented in which wake patterns are compared for airfoils plunging at different Strouhal numbers. In figure 2.15, the upper figure shows a Karman Vortex Street, where the rotation of the vortices above the middle line occurs clockwise and the rotation of those below the line is counterclockwise, therefore drag is produced (low Strouhal). On the

other side, the lower figure exhibits an opposite situation, indicative of thrust production (high Strouhal). These results are valid for viscous fluids, in which airfoils can produce drag when plunging at low Strouhal numbers. For inviscid flows plunging airfoils produce thrust at all Strouhal numbers, as it will be shown in the results section. This has to be taken into account when the thrust computed with the panel methods is compared with the one found experimentally.

Further experimental results are presented in [6, 10], where **tandem** and **ground effect** configurations are tested and the advantages over the single airfoil flapping are explained. A brief explanation is required at this point on what is meant by the use of ground effect, that is conventionally used for wings that travel in close proximity of a solid surface. In the theory of potential flow the configuration of a vortex next to a wall is often simulated by placing another vortex in a symmetric position with respect to the wall. This makes sure that the impermeability condition on the wall is fulfilled and that only the tangential velocity is left. Therefore from a numerical point of view to use two vortices in the flowfield is the same as to use a vortex and a solid surface. As an extension to this, two wings flapping symmetrically in a **biplane configuration** have been indicated as ground effect configuration in the remainder of the thesis.

One of the first experimental works carried out on bird-like wings has been presented on Nature in 1979 by N.V. Kokshaysky [93], in which he takes snapshots of a bird during its motion in a particle

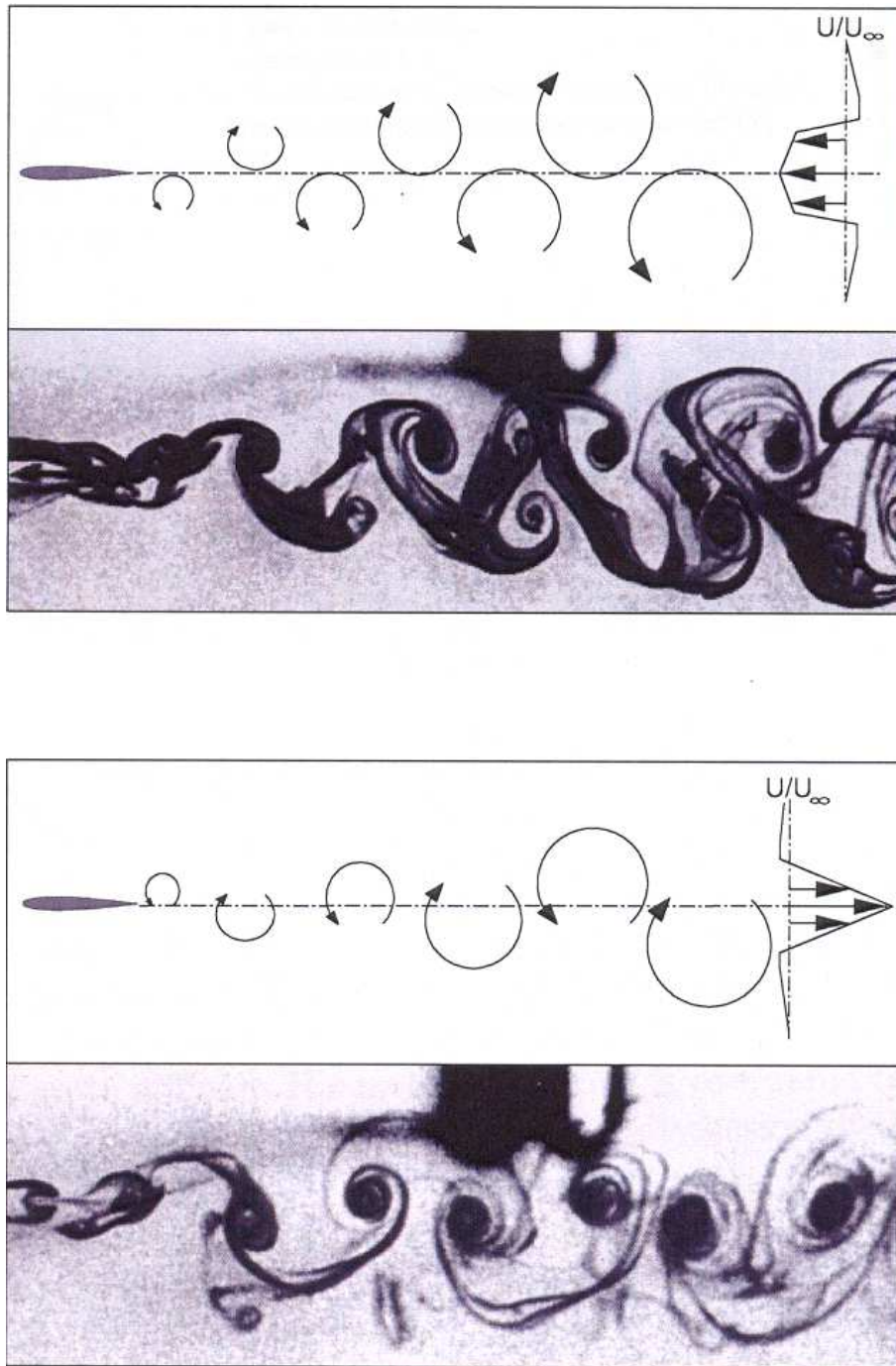


Figure 2.15: Karman Vortex Street (up) and Reverse Karman Vortex Street (down), from [8]

cloud. The observations led to the idea, commonly accepted, that the wake of birds is built up of *vortex rings*, with the upper part formed by the starting vortices of both the wings at the beginning of the downstroke. The lower parts consists of the stopping vortices at the end of the downstroke. These two vortices are closed against each other at the beginning and the end of the downstroke. Furthermore, all the birds seem to exploit only the downstroke in order to generate lift and propulsion, whereas in the upstroke their wings are are folded and they do not interact significantly with the air.

The work of Willmott, Ellington and Thomas [31, 94, 95, 96, 97] and Dickinson ([98, 99]) is noteworthy, because of the presence of the leading edge vortex they observed on the wing of a bird. They have also reproduced the experiment in the wind tunnel, using smoke to visualize the pattern of the flow over the wing. The wind tunnel results confirm the separation of the flow at the leading edge of the wing, but the region where the vortex is released from the wing lies at 75% of the wingspan. Another important feature of the vortex is the outward speed of its core, confirmed in the same experiment from the smoke direction: the smoke lead is placed next to the wing root and the separation of the flow is next to the wing tip.

Lin and al. [100] performed wind tunnel tests to measure the forces produced by a mechanical membrane flapping wing under different frequencies, angles of attack and speeds. The wing surface was formed by covering the composite skeleton with thin plastic film. The trends

of the lift and thrust are observed varying the aerodynamic parameters described above. The different behaviours of the root and tip sections are recorded: the sections closer to the root have very low flapping speed contribution, therefore all the lift is generated by the flying speed. The sections next to the tip on the other hand have flight and flapping speeds comparable between each other, with the consequence that a greater angle of attack is present.

2.4 Aeroelasticity of Flapping Wings

In the design of a fixed or rotary wing aircraft, the unsteady aerodynamic forces interacted with the wing vibration are normally treated as a negative factor as it may lead to disastrous consequences. The interaction of fluid and structure is one of the major concerns in aircraft structure design. Therefore efforts have been made to prevent excessive aeroelastic deformation under design requirements. Aeroelastic tailoring of composite wings is an approach to achieve a minimum mass optimal wing structure in terms of stiffness, inertia and the deformation shape [101, 102].

In the design of a flapping wing however, the unsteady aerodynamic force plays a more positive and beneficial role rather than a negative factor. In fact, it plays a significant role in generating the required lift and thrust from a flapping wing. Therefore the study of flapping wing aerodynamics has been attracting research attentions for more than

a decade [40, 103, 7, 8]. In this dissertation the attention is focused on two objectives. Firstly develop an aerodynamic model based on unsteady panel method as a flapping wing analysis tool; secondly investigate the aeroelastic effect on the aerodynamic forces produced by a flexible wing flapping at different frequencies especially near its resonance. This issue is particularly important for flapping wings because the aeroelastic deformation could be beneficial to enhance the forces on top of those produced by a prescribed rigid body flapping motion so as to increase the aerodynamic efficiency, as shown in [104, 103]. The research is aimed at designing an aeroelastic flapping wing of minimum weight and maximum efficiency by tailoring the flapping mode and the flexible wing structure.

The aeroelastic coupling is studied by solving Lagrange's equations of motion for a 2 degree of freedom (2-DOF) spring-mass wing section system. In the aeroelastic equations, the unsteady aerodynamic term is treated as an external force containing two parts. One of them is due to a specified rigid body flapping motion and another self excited part dependent upon the elastic displacements and their derivatives. If the flapping motion is ignored, the equations represent an aeroelastic stability problem and the flutter velocity can be obtained from the solution. If the flapping motion is included, the equations represent an aeroelastic dynamic response problem. In the first step of solving the problem, an initial aerodynamic force from the rigid body flapping motion is calculated as an external force in the equation. In the following step, the elastic displacements of the airfoil can be ob-

tained separately as an output of the equations and added on top of the rigid ones to give the total displacements. The total displacements (and their derivatives) are then passed to the panel method to calculate updated forces produced by the flexible wing. The procedure is conducted in an iterative manner in time domain, as shown in ([105, 106]).

Jones and Platzer [3, 4] spent a considerable amount of time in the investigation of the benefits that the deformation of the wing might have on the forces produced. In [3, 4] results are shown to assess the interaction of the wake released from a leading airfoil undergoing a flapping motion with a trailing airfoil with finite stiffness. The computations prove that the motion of the leading airfoil is beneficial for the aeroelastic stability of the trailing airfoil. Furthermore, the analysis of airfoils in ground effect shows the stabilizing effect of airfoils flapping close to the ground. Vehicle designed for low level flight benefit of this stabilizing effect when they fly three wing chords or less far from the ground. The beneficial effect, in both the configurations described, decays rapidly with the distance between the airfoils. Therefore the flutter control may be achieved with the placement of a canard or leading edge flap next to the airfoil. M.J. Smith [107, 108] presents a complete structure-aerodynamic analysis in his PhD dissertation, where a panel method is developed and coupled with a structural model of the wing to assess the advantage of the wing deformation. Attention is focused on the development of the finite element model, the panel method for the force computation and the Aerodynamic

Load Transformation Matrix. Different discretization procedures are employed by the structural and aerodynamic solvers. When the coupling occurs, the loads computed on the aerodynamic grid must be transformed onto the structural one, paying attention to the conservation of the fundamental properties of the load system. In the same manner, the structural displacements must be transformed onto the aerodynamic control points, and this can be achieved using the same transformation matrix. Muniappan and al. [109, 110] compares the lift of a flapping wing with torsional flexibility with a rigid flapping wing. The torsional flexibility is achieved fabricating the wing with two-flap arrangement. Therefore the wing can be used as rigid, single flap and double flap by arresting the motion of the flap. The results obtained show that the lift is the highest in the double-flap configuration, about 1.5 times the lift of the rigid wing. The arrangement is used in stick-free condition, therefore no input is given to the flaps. Jeff Eldredge and al. [71, 68, 111] investigates the coupling of the motion of flapping airfoils and the airflow in the surrounding, producing results that compare reasonably well with experiments carried out in a water tank. For a two-airfoil arrangement, the leading airfoil is given a prescribed motion, while the trailing body motion is passive. The deflection angle of the trailing body is related to the moment exerted by the fluid on the hinge. The algorithm is built up of two main steps:

1. *Fluid Evolution*: The strength and position of the vortices in the field are updated based on the Vortex Particle Method.

2. *Body Evolution*: The equations describing the body dynamics are integrated. The hinge angle is found iteratively at the end of the time step.

Investigations on a three-linkage 'fish' undergoing undulatory mechanics are presented in [68], where the results show that a reverse Karman vortex street is obtained. The alternating sign vortices released at each step indicate that thrust is being produced. The effectiveness of this model depends on the fluid viscosity, that acts as a dissipative mechanism. Shukla and Eldredge [111] present an extension of the model given in M.A. Jones [54], incorporating flexibility in order to assess the advantage of a flexible flat plate over a rigid one. The deformation of the body is prescribed on top of the plunge and pitch rigid motions, therefore there is no proper coupling between structure and airflow. However, the results seem to confirm the ability of the present method to incorporate flexibility in the computations.

Thanks to the increase of computer power, the analysis of the interaction of flow and structure has become possible using CFD solvers as well, as demonstrated in Silva ([112]) and Lisandrin ([113]). They used a reduced order modelling approach to compute the flutter boundaries of fixed wings in transonic flows, coupling a CFD solver (e.g.: FLUENT) and a finite element solver (e.g.: NASTRAN). Another example in which fluid-structure interaction is carried out with CFD methods is given by Khan ([114]).

Chapter 3

Methodology and Analysis

This chapter is dedicated to exposing the theoretical and numerical methods used in the rest of the thesis. Following the process outlined in the literature review, chapter 2, the current chapter will be explaining first the methods based on potential flow aerodynamics, and then will extend the discussion to the more general Vortex Particle Methods, that can be seen as a generalization to potential flow methods.

3.1 Theodorsen Theory

The theory of Theodorsen is based on the assumptions of **flat plate, small oscillations, sinusoidal input and inviscid flow**. The non stationary flow around a flat plate sinusoidal oscillations is considered, in order to obtain the forces acting on the plate when the wake effect is taken into account. Until the analysis of Theodorsen was published,

the analysis of unsteady flows was accomplished using the quasi steady theory, which is unsuitable for such a study. The derivation of the formula is not attempted in this contest, as no further development to the original theory has been achieved.

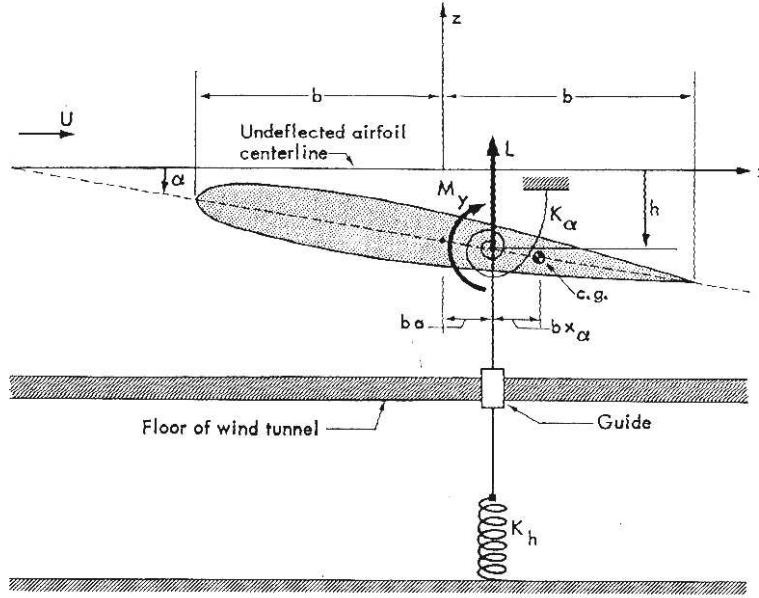


Figure 3.1: Semi-rigid airfoil with concentrated mass and stiffness undergoing small sinusoidal oscillations

Figure 3.15 illustrates a simplified model compared to the one from which Theodorsen theory is derived. The model shown in the figure has 2 degrees of freedom, whereas the model presented in Theodorsen has 3: heaving h , pitching α and aileron β . The degrees of freedom considered in this thesis are heaving and pitching. The formulae expressing the unsteady lift and pitching moment about the elastic centre on the flat plate are:

$$L = \pi \rho b^2 [\ddot{h} + U \dot{\alpha} - b a \ddot{\alpha}] +$$

$$+2\pi\rho UbC(k) \left[\dot{h} + U\alpha + b\left(\frac{1}{2} - a\right)\dot{\alpha} \right] \quad (3.1)$$

$$\begin{aligned} M = & \pi\rho b^2 \left[ba\ddot{h} - Ub\left(\frac{1}{2} - a\right)\dot{\alpha} - b^2\left(\frac{1}{8} + a^2\right)\ddot{\alpha} \right] + \\ & +2\pi\rho Ub^2\left(a + \frac{1}{2}\right)C(k) \left[\dot{h} + U\alpha + b\left(\frac{1}{2} - a\right)\dot{\alpha} \right] \end{aligned} \quad (3.2)$$

where U is the freestream velocity, ρ is the air density, b is half the airfoil chord, a is the distance of the pivot from the centre of rotation divided by b and $C(k)$ is a complex function of the reduced frequency k . Theodorsen identified this function as a combination of Hankel functions of the second kind, as given in the following formula:

$$C(k) = F(k) + iG(k) = \frac{H_1^{(2)}(k)}{H_1^{(2)}(k) + iH_0^{(2)}(k)} \quad (3.3)$$

Equation 3.3 separates the real and imaginary parts of the function $C(k)$, which will be used later for further derivation. If the apparent mass term is neglected in the formulae, then the forces consist of a function of $\alpha, \dot{h}, \dot{\alpha}$ multiplied by the complex function $C(k)$. For $k \leftarrow 0$ the function C is real and equal to 1. If the motion consists of pure plunging, then the only term present is $\dot{h}$, and the quasi steady lift is proportional to this vector, with the same direction and opposite phase: the lift is therefore a damping force opposing h . When the assumption of quasi steady aerodynamics does not hold, $\dot{h}$ is multiplied by a complex function: the result is that the magnitude is lower, as the magnitude of $C(k)$ is less than 1, and phase shift is negative, which makes the complex lift lie behind its quasi steady counterpart, as illustrated in figure 3.2.

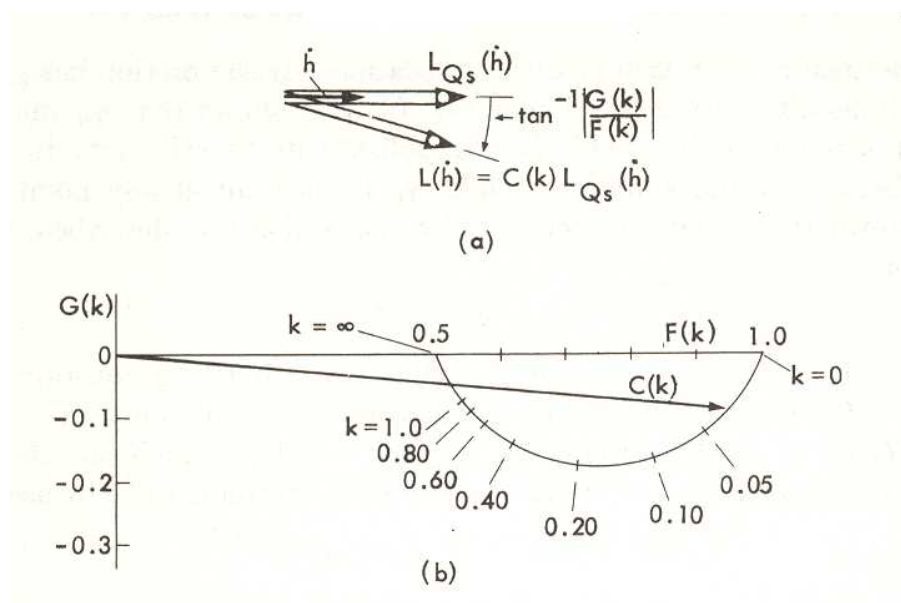


Figure 3.2: Sketch explaining the meaning of $C(k)$ in the complex plane

The equations 3.1 are not developed further in the thesis. They have been used as a means of comparison for when experimental results were not available. It is important to outline that Theodorsen formulae as written above are given in hybrid form, since the derivatives are respect to time, whereas the function $C(k)$ is in frequency domain. Since $C(k)$ does not have an analytic expression in time domain, the correct way to write 3.1 is to turn the $h, \alpha, \dot{h}, \dot{\alpha}, \ddot{h}, \ddot{\alpha}$ into frequency domain. In this thesis the hybrid notation has been used, because it is the most widespread across the books of Aeroelasticity and Unsteady Aerodynamics. The reader is warned to consider the formulae with particular attention when wondering the parameters needed to start the computation. The theory given in Theodorsen and expressed in the form of equations 3.1 is referred to as *Linear Theory* in the rest of the dissertation, for the underlying assumptions. To complete the

framework that will be used in the following chapters, the extension accomplished in [36] is quickly presented below. Garrick extended Theodorsen's work in order to obtain the propulsive force produced by a flat plate undergoing small amplitude oscillations. Following the method of Von Karman and Burgers, the thrust produced by the flat plate is computed balancing the work done to maintain the oscillations with the energy increase in the wake and the work done by the propulsive force. The propulsive force is expressed in terms of its average value over a period of oscillation and it is presented here for the particular case of one-degree-of-freedom oscillations. If the motion of the airfoil consists of heaving with amplitude h_0 and reduced frequency k computed using the half chord as the reference length, the average thrust coefficient is:

$$C_T = \pi \left(\frac{\omega b}{U_\infty} \frac{h_0}{b} \right)^2 (F^2 + G^2) = \pi (k \hat{h})^2 (F^2 + G^2) \quad (3.4)$$

where $\hat{h}$ is the non-dimensional amplitude of oscillation. The average thrust coefficient is an important parameter that will be used to compare the performance of flat-plate/flat/wake and thick-airfoil/deforming-wake. Another relevant parameter is the propulsive efficiency, which is given below for the plunge case again:

$$\eta = \frac{TU_\infty}{L\dot{h} + M\dot{\alpha}} = \frac{F^2 + G^2}{F} \quad (3.5)$$

The values of thrust coefficient and efficiency expressed by equations

3.4 and 3.5 are ideal values, to which the ones obtained in the numerical computations using different tools will be compared for validation. For example, the value of the efficiency for zero frequency oscillations (e.g.: for oscillations with a very small reduced frequency) is 1, and this is due to the assumption that the wake behind the plate is narrow. If this assumption is removed, the efficiency will be not unitary anymore, due to energy lost in the shedding process.

3.2 Unsteady Aerodynamics of Flat Plate Undergoing Large Amplitude Oscillations

A second algorithm has been considered for the analysis of the large amplitude motion of a flat plate. The model is based on the work of J. De Laurier and J. Fairgrieve [40], in which the theory of Theodorsen is extended to the case of a flat plate undergoing pitch and plunge large amplitude oscillations. The method is based on *Kelvin's theorem*, which states that no change in circulation occurs in time for an irrotational system. Therefore, if a change in circulation occurs on the airfoil, further circulation is shed in the wake to make the sum zero. As the circulation on the airfoil depends mainly upon the effective angle of attack, a time step procedure is easily implemented to calculate the intensity of the vortex elements shed in the wake at each time step. Due to the sharp trailing edge, an airfoil creates circulation around itself when it is moving. The total bound vorticity is built up of two main contributions: quasi-steady vorticity γ_0 , due to the

unsteady motion, and unsteady vorticity γ_1 , due to the influence of the wake elements on the airfoil. As shown in [40], the quasi steady contribution is given by:

$$\gamma_0(x) = 2U\alpha' \sqrt{\frac{1-x}{1+x}} + 2\dot{\alpha} \sqrt{1-x^2} \quad (3.6)$$

where α' is the effective angle of attack defined by:

$$\alpha' = \sin \alpha + \frac{\dot{h}}{U} \cos \alpha \quad (3.7)$$

The bound vorticity induced by the wake on the airfoil is computed using the expression:

$$\gamma_1(x) = \frac{1}{\pi} \sqrt{\frac{1-x}{1+x}} \int_1^\infty \frac{\gamma_w(\xi)}{\xi-x} \sqrt{\frac{\xi+1}{\xi-1}} d\xi \quad (3.8)$$

where the integral's lower limit is the trailing edge of the airfoil. At the first time step, the intensity of the vortex released in the wake is equal in value and opposite in sign to the circulation developed on the airfoil; in the following time steps the last element shed is computed enforcing the zero circulation condition, considering all the elements shed in previous time steps.

$$\Gamma_j^w = -\sqrt{\frac{\xi_j-1}{\xi_j+1}} \left(\Gamma_0(t=j\Delta t) + \sum_{k=1}^{j-1} \Gamma_k^w \sqrt{\frac{\xi_k+1}{\xi_k-1}} \right) \quad (3.9)$$

where:

$$\xi_k = 1 + (j + 1 - k)U\Delta t \quad k = 1, 2, \dots, j$$

is the position of the generic vortex with respect to the leading edge, with the semi-chord of the airfoil equal to 1. Equation 3.9 is valid for a planar wake, since it takes into account only the position of the vortices on the airfoil axis; a more complicated expression is given for non planar wake in [40], and it is based on a conformal transformation of the airfoil onto a circle. It has been already pointed out the necessity of accounting for the real shape of the wake for an accurate solution of the problem. In a planar wake solution, as in Theodorsen, the thrust is not predicted since the force acting on the airfoil is normal to the wake elements. A suction force theory can predict the existence of an horizontal force, as in [36].

Fairgrieve and DeLaurier [40] present a method in which the the thrust is given by the projection of the resultant force onto the freestream velocity direction. Define the impulse of a vortex pair as the cross product of the circulation and the distance between the vortices, with the air density being a proportionality factor:

$$\mathbf{I} = \varrho \mathbf{x} \wedge \mathbf{\Gamma} \quad (3.10)$$

Then equation 3.10 states the presence of a force perpendicular to the wake as a generalization of the *Kutta-Joukowski theorem*. The lift and the thrust acting on the airfoil can be obtained from the equation

of the impulse by means of derivation in the directions respectively perpendicular and parallel to the freestream velocity:

$$T = \frac{d}{dt}(I_x) \quad (3.11)$$

$$L = \frac{d}{dt}(I_y) \quad (3.12)$$

The method presented is able to account for unsteady effects of the wake without the introduction of any function of the frequency (see Theodorsen), therefore no quasi-steady assumption is made, other than the one used to compute the effective angle of attack on the plate. Three different versions of the method are developed, based on the way the wake is computed:

- *Planar Wake* : as in Garrick and Theodorsen;
- *Frozen Wavy Wake*: the wake elements remain *frozen* in the positions in which they have been shed from the trailing edge;
- *Deformable Wavy Wake*: wake elements are free to move under the influence of themselves, the other elements and the airfoil.

Although the second scheme doesn't fulfil the equations of fluid motion, it is more accurate than the first one when amplitudes become large, as it accounts for a more realistic path of the wake than the planar one. The third scheme is the most complicated, because at the

end of each time step, after computation of the circulation, it requires to update the position of the vortices before starting a new time step. If we define coefficients A and B as:

$$A = \int_{-1}^1 \gamma(x) x dx + \int_{wake} \gamma_w(s) \xi ds$$

$$B = \int_{wake} \gamma_w(s) \eta ds$$

it can be shown that B accounts for *out-of-plane* displacement of the wake and it equals zero for the planar wake model. The impulses in the x and y are given by the integrals:

$$I_x = -\varrho \int_{vorticity} \gamma(s) Y ds$$

$$I_y = \varrho \int_{vorticity} \gamma(s) X ds$$

If we employ the airfoil's coordinate system and take the origin at midchord, it follows:

$$x = (X - X_0) \cos \alpha - (Y - Y_0) \sin \alpha \quad (3.13)$$

$$y = (X - X_0) \sin \alpha + (Y - Y_0) \cos \alpha \quad (3.14)$$

Substituting 3.13 into the impulse equation, it can be finally written:

$$I_x = \varrho(A \sin \alpha - B \cos \alpha) \quad (3.15)$$

$$I_y = \varrho(A \cos \alpha + B \sin \alpha) \quad (3.16)$$

Derivation of these expressions yields:

$$T = \varrho(\dot{A} + B\dot{\alpha}) \sin \alpha + \varrho(A\dot{\alpha} - \dot{B}) \cos \alpha \quad (3.17)$$

$$L = -\varrho(\dot{A} + B\dot{\alpha}) \cos \alpha + \varrho(A\dot{\alpha} - \dot{B}) \sin \alpha \quad (3.18)$$

The moment of momentum acting on the airfoil can be obtained in a similar way. Although a flat plate model does not have any practical application, due to the fact that wings are cambered and have a thickness different from zero, this method is useful to understand the complicated mechanisms underlying combined pitch and plunge oscillations. The generation of the thrust is due to wake elements interaction and out-of-plane motion, which is a characteristic of flapping airfoils. It is interesting to compare this model with Theodorsen theory, to assess the contribution of the wake deformation to the forces achieved. In the limits of small oscillations, both the models yield the same results. For larger amplitudes the model of Theodorsen fails to capture the real trend of the forces for it is limited to small oscillations. Therefore the model presented in [40] should produce more accurate estimates of the aerodynamics. It is very important and of practical interest to understand the differences between the Frozen Wavy wake and the Deformable Wavy Wake models: can the wake

deformation play a relevant role in the generation of lift and thrust or is its effect neglectable in the conditions we are interested? It is well known that the wake deformation has an effect on the efficiency, as energy is spent for the generation and the displacement of the vortices. The higher the frequency of oscillation, the greater the amount of energy dissipated in the shedding process. The reason for which the Frozen and Deformable Wake are considered is because this effect needs to be quantified to understand if it is worth taking it into account.

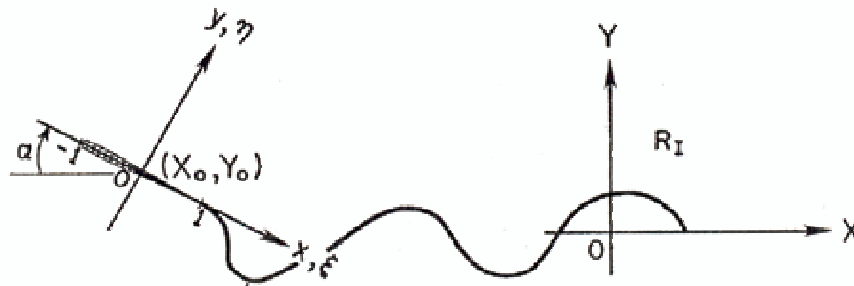


Figure 3.3: Change of coordinate system: from body to inertial

3.3 Panel Methods

Physically, the assumption of potential flow cannot strictly represent the flow field on a flapping wing, as vorticity is released from the leading and trailing edge of the airfoil, whereas the main characteristic of potential flows is irrotationality. For our purpose, let us consider the incompressible flow around an arbitrary body. At this point there

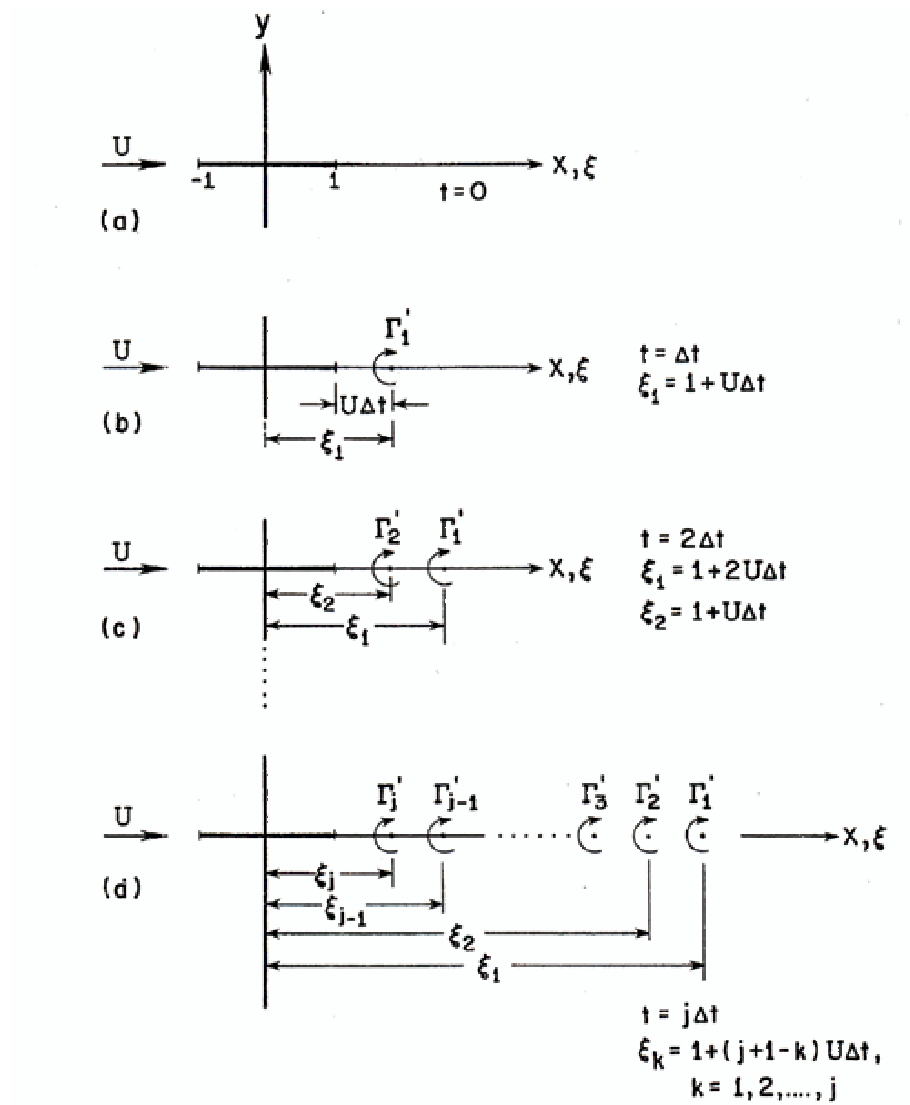


Figure 3.4: Vortex generation and convection in a planar wake model

is no need to specify either the body shape or the dimension of the domain. If the mathematical model employed to describe the physical phenomena occurring in the domain is the *Potential Flow*, the assumptions on which it is based are:

- *Zero Viscosity*;
- *Irrotationality for $t \geq 0$* ;
- *Flow Attached on the Body Surface*;

Before developing the framework that is going to be used for the analysis, we want to show that the assumptions made are valid for the treatment of the flow field around flapping wings. Let us first define the *circulation* as

$$\Gamma = \oint_C \mathbf{v} \cdot d\mathbf{x} = \Gamma(t) \quad (3.19)$$

then *Kelvin theorem* says that the circulation around a closed curve C built up of the same fluid elements is constant in time. This can be also expressed saying that the rate of change of circulation in time is zero, or that the derivative of the circulation is zero:

$$\frac{d\Gamma}{dt} = 0 \longrightarrow \Gamma(t) = \Gamma(0) \quad (3.20)$$

For proof of equation 3.20 see [12], pp. 25-26. The circulation is an integral quantity, therefore assuming that the circulation is constant

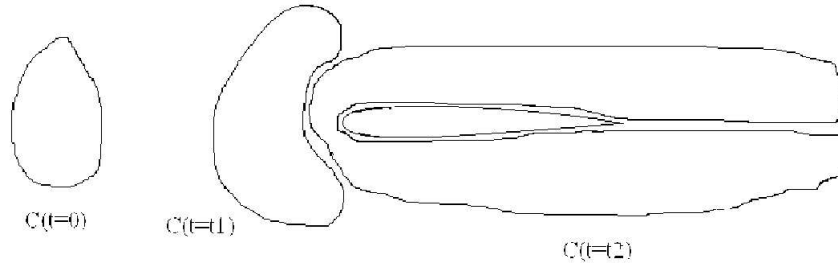


Figure 3.5: Illustration of material curves encircling the body

does not mean that the vorticity is zero in each point of the flowfield. Figure 3.5 illustrates the motion of a material curve in time. At $t = t_2$, the curve surrounds the body, that is the boundary of the domain in which the material elements lie. Applying Kelvin's Law to the curves C_0, C_1 and C_2 and making use of the *Stokes theorem* yields:

$$\Gamma(t) = \oint_C \mathbf{v} \cdot d\mathbf{x} = \int \int_S \boldsymbol{\omega} \cdot \mathbf{n} dS = 0 \quad (3.21)$$

where

$$\boldsymbol{\omega} = \nabla \wedge \mathbf{v}$$

This leads to the conclusion that in a bounded domain, a scalar function can be defined whose gradient is equal to the velocity field:

$$\mathbf{v} = \nabla \Phi \quad (3.22)$$

The function Φ is the total *Potential* and the way it is defined satisfies automatically the Navier Stokes equation for inviscid flows. Replacing the velocity vector with the gradient of the potential in the mass conservation equation for incompressible flows yields:

$$\nabla^2 \Phi = 0 \quad (3.23)$$

with the boundary conditions on the body surface and at the farfield (see [12]):

$$\nabla \Phi \cdot \mathbf{n} = 0 \quad (3.24)$$

$$\nabla \phi(\mathbf{x} \rightarrow \infty) = 0 \quad (3.25)$$

with

$$\Phi = \Phi_\infty + \phi$$

The first boundary conditions indicate that the component of the gradient of the total potential normal to the body surface has to be zero; the second boundary condition requires that the flow disturbance should diminish far from the body. Under the assumptions described above and releasing the hypothesis that the punctual vorticity must be zero, the flow is called *Quasi-Potential*. The only points in which the flow is irrotational belong to a surface (line in 2D) that represents the boundary of the potential function, along with the body and the freestream. The equations that hold across the discontinuity surface

are the conservation of mass and momentum, given as jump relations between the two sides of the surface:

$$\Delta [\rho (v_N - v_S)] = 0 \quad (3.26)$$

$$\Delta [\rho (v_N - v_S) \mathbf{v} + p \mathbf{n}] = 0 \quad (3.27)$$

In the equations above the v_N represents the normal component of the velocity of the particles adjacent to the surface on the upper and lower sides, while v_S indicates the velocity of the surface. From the jump relations, after passages skipped in this derivation and that can be found in [43], we obtain:

$$\Delta v_N = 0 \quad (3.28)$$

$$\Delta p = 0 \quad (3.29)$$

$$v_N - v_S = 0 \quad (3.30)$$

The above derivations show that the particles adjacent to the surface have the same normal component of the velocity as the surface, that is not crossed by particles. Besides the surface has no pressure jump. The wake elements in the flowfield at the generic time t represent particles that have been shed in the previous time steps. Their strength can be found using the unsteady expression of Bernoulli's theorem across the wake. The unsteady expression of Bernoulli can be thought of as an integral of the Euler equation for the momentum when no vorticity

is present in the flowfield. When the Bernoulli equation is applied on each side of the discontinuity surface and the two equations are subtracted from each other, considering that no pressure jump exists across the wake, we have:

$$(\dot{\Phi}_u - \dot{\Phi}_l) + \frac{1}{2} (|\nabla\Phi_u|^2 - |\nabla\Phi_l|^2) = \quad (3.31)$$

$$\Delta\dot{\Phi} + \left(\frac{v_u + v_l}{2} (v_u - v_l) \right) = \quad (3.32)$$

$$\dot{\Gamma} + v_W \cdot \nabla (\Gamma) = \quad (3.33)$$

$$\frac{D_w \Gamma}{Dt} = 0 \quad (3.34)$$

where Γ indicates the strength of the element shed in the wake.

In the previous derivation it has been assumed that the wake velocity is equal to the average of the velocity of the fluid particles on the lower and upper sides of the wake surface. The final result is that the elements leaving the airfoil does not change its strength during the convection process. Hence the only unknowns at each time step are the doublets on the body and the last wake element shed.

The solution of equation 3.23 with appropriate B.C. is obtained by a boundary element method, in which only the boundaries of the field are discretized. Following *Green's identity*, a solution can be built distributing singularities with unknown strength on the body and the wake. The general solution is given in the form:

$$\Phi^* = \frac{-1}{4\pi} \int_{S_B} \left[\sigma \left(\frac{1}{r} \right) - \mu \mathbf{n} \cdot \nabla \left(\frac{1}{r} \right) \right] dS + \Phi_\infty \quad (3.35)$$

where

$$\Phi_\infty = U_\infty x + V_\infty y + W_\infty z$$

is the potential associated with the freestream velocity. For a complete derivation of the method see [12]. Deriving both sides of equation 3.35 with respect to the vector normal to each panel of the body we obtain the condition of impermeability onto the body surface, also known as the Neumann boundary condition:

$$\left[\sum_{j=1}^{N_P} \frac{1}{4\pi} \int_{B_j} \mu \nabla \frac{1}{r} dS + \sum_{k=1}^{N_W} \frac{1}{4\pi} \int_{B_k} \mu \nabla \frac{1}{r} dS - \sum_{j=1}^{N_P} \int_{B_j} \sigma \frac{1}{r} dS \right] \cdot \mathbf{n} = 0 \quad (3.36)$$

If the body surface and the wake get split into N panels, each being associated with a source and a doublet element, a system of $2N$ unknowns will be generated. Since the boundary condition applied on the collocation points placed over each panel can provide N equations, either the doublet's or the source's strengths must be assigned. Conventionally the strength of the sources is equalled to the normal wash on each panel:

$$\sigma_i = \left(\frac{\partial \Phi}{\partial n} \right)_i = (\mathbf{v}_B \cdot \mathbf{n})_i \quad (3.37)$$

Moreover, at each time step a wake panel is generated, due to the element leaving the trailing edge; thus the system to be solved includes also the contribution of the wake panels shed at the current and at previous time steps. After discretization the system becomes:

$$\sum_{j=1}^{N_P} C_{ij} \mu_j + \sum_{k=1}^{N_W} C_{ik} \Gamma_k + \sum_{j=1}^{N_P} B_{ij} \sigma_j = 0 \quad i = 1, N_P \quad (3.38)$$

The last wake element shed has unknown strength and direction, as it depends on the fluid velocity and the circulation on the airfoil at the present time step. Platzer and Jones [4] employ an iterative scheme with which both strength and direction are treated as unknowns. The solution is obtained when the direction and the strength found result in a zero pressure jump at the trailing edge. In the present thesis, the direction is assumed known and equal to the direction of the airfoil's speed at the trailing edge. The strength of the last wake panel shed is given using one of the numerical expressions of the Kutta condition, as in [12].

$$\Gamma_W = \mu_{N_P} - \mu_1 \quad (3.39)$$

If the source and the wake terms are brought to the right hand side, and the contribution of the last wake panel is included in the unknowns vector, we finally have the $(N + 1) \times (N + 1)$ linear system:

$$\begin{bmatrix} a_{11} & a_{12} & \cdots & a_{1N_P} & a_{1N_W} \\ a_{21} & a_{22} & \cdots & a_{2N_P} & a_{2N_W} \\ \vdots & \vdots & \ddots & \vdots & \vdots \\ a_{N_P1} & a_{N_P2} & \cdots & a_{N_PN_P} & a_{N_PN_W} \\ -1 & 0 & \cdots & 1 & -1 \end{bmatrix} \begin{pmatrix} \mu_1 \\ \mu_2 \\ \vdots \\ \mu_{N_P} \\ \Gamma_{N_W} \end{pmatrix} = \begin{pmatrix} RHS_1 \\ RHS_2 \\ \vdots \\ RHS_{N_P} \\ 0 \end{pmatrix} \quad (3.40)$$

which can be solved with the traditional methods of linear algebra. After having found the strength of the doublets, the induced velocity over the wing panels centroids can be computed.

The values found represent **perturbation velocities**, and can be used for the computation of the pressure coefficients. If Q_i is the total tangential velocity over the i^{th} panel and v_∞ is the freestream velocity, it can be written:

$$C_p = \frac{p - p_{ref}}{\frac{1}{2}\rho v_{ref}^2} = 1 - \frac{Q^2}{v_{ref}^2} - \frac{2}{v_{ref}^2} \frac{\partial \Phi}{\partial t} \quad (3.41)$$

The last step to be performed is the convection of the wake elements. Again, deriving the discretized Green's identity with respect to x, y yields the velocity of the flow in correspondence of each panel.

$$v_k^{x,y} = \sum_{m=1}^{N_P} b_{k,m}^{x,y} \sigma_m(t) + \sum_{m=1}^{N_P} c_{k,m}^{x,y} \mu_m(t) + \sum_{n=1}^{N_W} w_{k,n}^{x,y} \Gamma_{Wn}(t) \quad (3.42)$$

where the superscript x, y indicates the variable with respect to which the derivative is computed. The position of the wake elements can be updated in time using an explicit time integration scheme. In the

case of the panel method developed in this thesis the Euler scheme is used, as the time step is can be considered as small as needed to achieve a good accuracy. If greater accuracy is needed, the step can be performed with a 4th order Runge Kutta scheme. In general, the update process can be described as below:

$$\mathbf{x}_k = \mathbf{x}_{k-1} + \mathbf{u}_k \Delta t \quad (3.43)$$

An important assumption that has been made in the derivation of the method is that the strength of the sources and doublets over the body and the doublets representing the wake are constant over the panel length. This kind of discretization is referred to as *zero order*, in contrast with other more accurate methods in which the singularities have linear or higher order trends along the panel. Besides, the value of the velocity given by equation 3.43 is computed in the middle of the panel. This may generate numerical problems in the implementation of the method, as it will be described in the next section.

3.4 Extension of Panel Methods to Multiple Airfoils

The theory derived in the previous section can be extended in order to allow the panel method to deal with multiple airfoil configurations. At this point it is not important to define the position of the airfoils relative to each other, neither is important to establish whether the

airfoils have different shapes. The only extra-assumption made with respect to the previous section is that two airfoils are present in the flowfield, some distance apart from each other, and that they are undergoing arbitrary oscillations with, in general, different frequencies and amplitudes. The equation describing the physical model is 3.23. The boundary conditions are different, because two bodies are present in the flowfield, therefore the impermeability boundary condition must be fulfilled on both the bodies. The unknowns are $2N$, equal to the number of doublets on the two bodies, and the equations are $2N$, such as the total number of panels. Applying Green's Identity in the case of two airfoils leads to the discretized set of equations:

$$\begin{bmatrix} A_{1,1} & A_{1,2} & a_{1,2N_P+1} & a_{1,2N_P+2} \\ A_{2,1} & A_{2,2} & a_{2,2N_P+1} & a_{2,2N_P+2} \\ -1 \cdots 1 & 0 & -1 & 0 \\ 0 & -1 \cdots 1 & 0 & -1 \end{bmatrix} \begin{pmatrix} \mu_1 \\ \mu_2 \\ \Gamma_{N_{W1}} \\ \Gamma_{N_{W2}} \end{pmatrix} = \begin{pmatrix} \mathbf{RHS}_1 \\ \mathbf{RHS}_2 \\ 0 \\ 0 \end{pmatrix} \quad (3.44)$$

In the set 3.44, the matrix A is has in general all the elements different from zero, with the exception of the last two rows, that represent the Kutta-Joukowski condition. Therefore the value of the doublets on each airfoil are dependent on the values on the other airfoil. Moreover, the wakes element in the field affect the values of the doublets on both the airfoils and also the computation of the velocity in the convection step. As the influence of doublets and vortices on the potential and the velocity in a given point decreases with the distance from the

point, it has to be expected that the closer the airfoils, the greater the cross-influence. In the limit of two bodies infinitely distant from each other, the set of equations would be *uncoupled* and the cross-influence would be non-existent. In the form adopted for the expression of the set of equations 3.44 it has been made use of blocks, in order to represent the influence of the airfoils on themselves ($A_{i,i}$), and the influence that the doublets of one airfoil have on the doublets of the other airfoil ($A_{i,j}, j \neq i$). The greater the distance between the airfoils, the smaller the coefficients of the anti-diagonal blocks. At this point of the discussion no assumption has been made on the relative motion of the two airfoils, which can both be flapping with different frequencies and amplitudes. In practice, we will narrow the field of study and restrict our investigation to two examples that are of great interest for applications in MAVs.

3.5 Panel Method Implementation

In this section details are given on how the panel method has been implemented in the thesis. The source code is written in Fortran 90 and compiled with the default compiler available on Linux (*pgf90*). The size of the code is such that it can be run on a desktop computer with a 2.8 GHz processor within a few minutes for each simulation. The input to the program is the shape of the airfoil, given as a set of points starting from the trailing edge, which appears twice in the input file in order to form a closed shape. The parameters that are

needed to start the computation, other than the shape, are the angle of attack, the speed of the flow, the amplitude and the frequency of the flapping motion. The frequency of oscillation is the same for pitch and plunge, but the amplitudes are in general different. Besides, the initial angle of attack can be set different from zero: this allows to compute the steady flow around an airfoil once the transient has vanished. The rotational motion of the airfoil is rigid, therefore the pivot position is input as well. The other parameters needed to start the computation are the time step and the number of steps performed. The time step is not chosen arbitrarily, but it is in a close relationship with the frequency of oscillation. From a theoretical point of view, the *Nyquist Law* states that two points per cycle are needed to capture a sine wave. Therefore, if f is the oscillation frequency, then the maximum time step size is given by:

$$\Delta t = \frac{1}{2f}$$

Although this is the greatest time step size allowed, the actual value of the time step is always set to at least one tenth of this value. This is to guarantee a certain smoothness in the computation of the forces and the wake. In particular, two more parameters have been introduced that represent the number of samples for each oscillation cycle (NSAMPLES) and the number of cycles performed in one computation (NCYCLES). The total number of time steps can be obtained by means of the product $NSTEPS = NCYCLES \times NSAMPLES$. A

convergence study has been made on the number of samples per cycle to be used and the value of 100 seems to yield reasonably accurate results. In this study the maximum value of the lift coefficient has been recorded and the value obtained with the number of samples equal to 100 is accurate enough for our purposes. The choice of the oscillation frequency and the number of samples per cycle leaves no room for the choice of the time step. If T denotes the period of oscillation and ω the angular frequency, then the time step is given by:

$$\Delta t = \frac{T}{N_{samples}} = \frac{2\pi}{\omega N_{samples}}$$

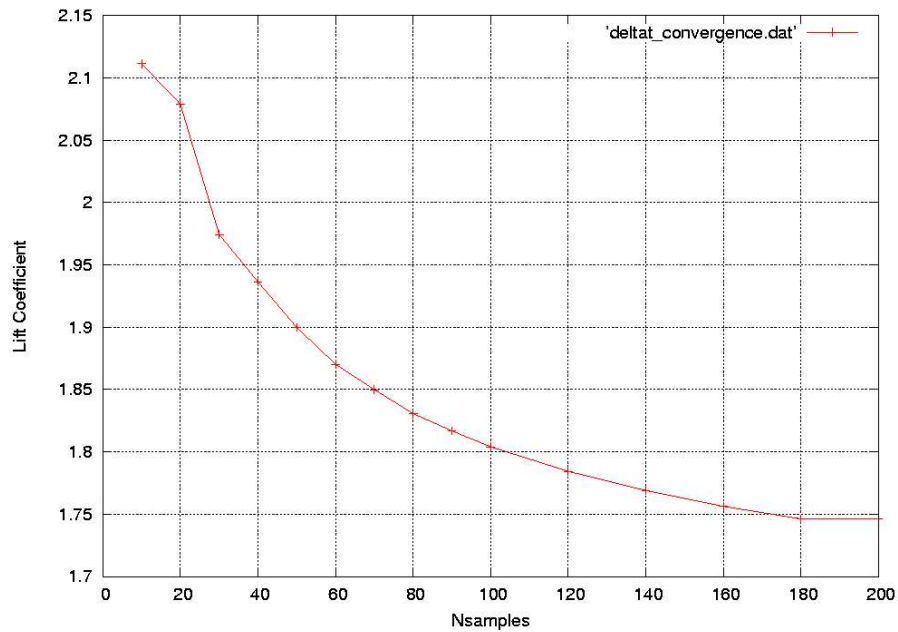


Figure 3.6: Convergence study on the Lift Coefficient with respect to Δt

Figure 3.6 shows the convergence study performed with respect to the number of samples, which is the same than saying the time step size. With 200 samples per cycle the value of the lift coefficient is captured

correctly, and this is used as the asymptotic value of the C_L . After the convergence study has been carried out and has shown that 200 time steps are required to have an accurate representation of the force coefficient, it has been decided to use 100 steps for the rest of the thesis. The percentage error that is made using 100 steps instead of 200 is less than 5%, as the graph highlights. The use of a smaller number of time steps improves the running time by a lot: the reason is that after a large amount of time steps has passed, most of the running time is due to the computation of the induced velocities on the vortices of the free wake. The solution of the linear system to find the doublets is independent upon the time step, as the size of the matrix to invert is fixed. The number of vortices (N) instead increases with time and this increases the time required to compute the induced velocities, which is proportional to N^2 . Therefore the reduction of time steps used can bring a great advantage with a small loss in accuracy. The modes considered for the oscillation are plunge and pitch. The first is such that all the points of the airfoil are displaced by an equal amount in the vertical direction. The latter is a rigid rotation about a pivot, as specified above. The function that defines the displacement of the airfoil in time is given in analytical form within the program. The other option is to input the amplitude of the displacement by means of a file. The function that is used for the computation is a sine (or cosine). Although the frequency of oscillation is the same for pitch and plunge, the phase between the two motions is different and the difference can be assigned arbitrarily. The output of the program are

the aerodynamic forces on the airfoil and the shape of the wake. The forces are output as thrust, lift and moment coefficients. In order to obtain these values the pressure coefficient on the airfoil's panels is integrated. Equation 3.41 shows the continuous form of the pressure coefficient, that needs to be discretized numerically. The velocity on each panel is obtained by means of equation 3.42. The component of the velocity we are interested in is the one tangent to the panel, therefore the x, y components obtained in 3.42 need to be projected onto the i^{th} panel via the scalar product with the panel versor. The numerical discretization of the time derivative of the potential does not represent a particular challenge. For the generic time step, it can be evaluated with a central finite difference formula:

$$\left(\frac{\partial \Phi}{\partial t}\right)_i^j = \frac{\Phi(i, j+1) - \Phi(i, j-1)}{2\Delta t}$$

where i denotes the panel and j the time step. For the first and last time step, a central difference formula cannot be used, and a first order formula is used instead. In particular, for the first time step an upwind formula is used, with the initial time step being assigned a zero potential, whereas the last time step is computed using a downwind formula. Once the pressure coefficient is found, it can be integrated to obtain the force coefficients. The thrust and the lift are found by projecting the pressure coefficient onto the freestream and the normal-to-freestream directions. The moment coefficient is found with respect to the quarter-of-chord point. First, the forces coefficients

are computed in the $\xi\eta$ coordinate system, where ξ is the tangent-to-chord direction and η is the normal-to-chord direction. The thrust, lift and moment coefficients in this coordinate frame can be written as:

$$\begin{aligned} c_\xi &= \sum_{i=1}^{N_P} c_{p,i} \Delta s_i \sin \theta_i \\ c_\eta &= \sum_{i=1}^{N_P} c_{p,i} \Delta s_i \cos \theta_i \\ c_{M,c/4} &= \sum_{i=1}^{N_P} c_{p,i} \Delta s_i (-\sin \theta_i (\xi_i - 0.25c) + \cos \theta_i \eta_i) \end{aligned} \quad (3.45)$$

where it has been used the fact that the quarter of chord lies on the ξ axis therefore $\eta_{c/4} = 0$. Due to the pitch motion, these coefficients are in general different from the lift and thrust coefficient. It is remarkable that the moment coefficient remains the same in both the airfoil and wind coordinate systems. The coefficients in the wind reference frame are given by:

$$\begin{aligned} c_L &= c_\eta \cos \alpha + c_\xi \sin \alpha \\ c_T &= -c_\eta \sin \alpha + c_\xi \cos \alpha \end{aligned} \quad (3.46)$$

The central point in the implementation of the panel method is the wake shedding and convection process. As it will be shown when comparing the results obtained with flat and free-to-evolve wakes, the free convection of the vortex elements yields results completely different from the flat wake assumption. In the panel method described

by Jones and Platzer the wake is assumed to be built up of point vortices carrying an amount of vorticity such that the Kelvin condition is fulfilled. In the present case the wake is built up of doublets, that need two points to be defined. In the *zeroth* order discretization the strength of the doublets is constant throughout the panel length. Once the doublet is shed off the body, the panel moves according to the velocity induced by the other singularities. The velocity is computed in correspondence of the panel mid-point and it is assumed to be constant over the whole panel. Let us consider two adjacent wake panels: before their positions are updated, the left vertex of the right element and the right vertex of the left element occupy the same positions. The velocities of the mid points of the two elements are in general different, and this will lead to the panels not being connected once their positions are updated. One way to overcome this problem is shown by Jones and Platzer and it consists of concentrating the vorticity in the middle of the panel and move the panel as if it were a point vortex. This is not straightforward when doublets are used to represent the body panels. Another way to get around the problem is to define two arrays, one with the left and one with the right vertices of the panels. When updating the position of the wake, the condition that the left vertices coincide with the right ones is imposed and the wake continuity is preserved. A first drawback of this method is the extra memory required for the storage of the same points twice, on the left vertex array and on the right vertex array. As it has been outlined above, memory and CPU time are not issues in our case. The

second drawback is that the panel stretches and shrinks depending on the positions of its vertices. Therefore it is not convenient to deal with panels when updating the wake position.

The novelty of the approach employed in the present dissertation consists in treating the wake elements as point vortices with strength equal to the difference between the circulation released at the generic time step and the previous time step. In a certain way the approach that is being described is similar to the one suggested in Jones and Platzer, except for the placement of the vortices in the flowfield. As it has been outlined above, the difference between a doublet and a vortex is that a vortex needs one point and the strength to be completely defined, whereas the doublet needs two points and the strengths of the vortices in these points.

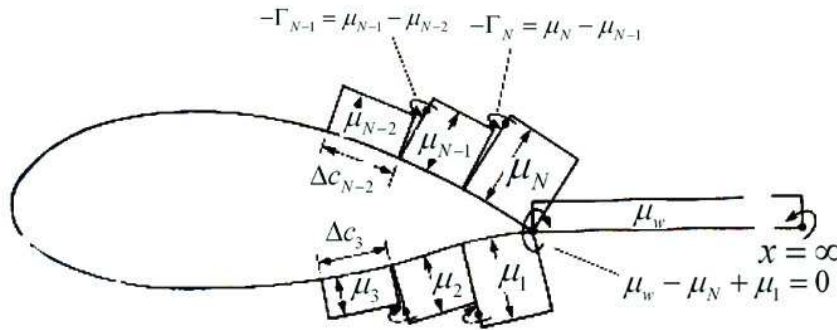


Figure 3.7: Convection of the wake in a steady flow

Figure 3.7 shows the wake after some time has elapsed from the shedding of the first element. Physically the wake is a continuous sheet. In the computation the steady state is achieved after a long time has passed and the Wagner effect of the initial vortex has vanished. At

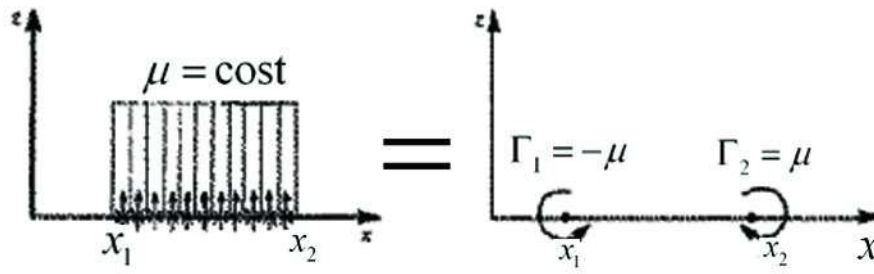


Figure 3.8: Equivalence between a doublet and a vortex pair

each time step, a doublet of vertices $P_1 = (x_1, z_1)$ and $P_2 = (x_2, z_2)$ and strength μ is released from the trailing edge and its influence over a point $P = (x, z)$ can be computed as:

$$\begin{aligned}
 \Phi_D &= -\frac{\mu}{2\pi} \left[\tan^{-1} \frac{z}{x - x_2} - \tan^{-1} \frac{z}{x - x_1} \right] \\
 u_D &= -\frac{\mu}{2\pi} \left[\frac{z}{(x - x_1)^2 + z^2} - \frac{z}{(x - x_2)^2 + z^2} \right] \\
 w_D &= -\frac{\mu}{2\pi} \left[\frac{x - x_1}{(x - x_1)^2 + z^2} - \frac{x - x_2}{(x - x_2)^2 + z^2} \right]
 \end{aligned} \tag{3.47}$$

As shown in [12], the influence of a vortex with strength Γ and position $P_0 = (x_0, z_0)$ over the same point P is:

$$\begin{aligned}
 \Phi_V &= -\frac{\Gamma}{2\pi} \tan^{-1} \frac{z}{x - x_0} \\
 u_V &= \frac{\Gamma}{2\pi} \frac{z}{(x - x_0)^2 + z^2} \\
 w_V &= -\frac{\Gamma}{2\pi} \frac{x - x_0}{(x - x_0)^2 + z^2}
 \end{aligned} \tag{3.48}$$

Comparison of equations 3.47 and 3.48 is straightforward and allows to conclude that a constant doublet is equivalent to a pair of vortices

placed $(x_2 - x_1)$ apart and with strengths $\Gamma_1 = -\mu$ and $\Gamma_2 = \mu$. Going back to the steady case discussed above, it is possible to represent the continuous wake sheet with two vortices whose positions are $P_1 = (x_{TE}, z_{TE})$ and $P_2 = (\infty, z_{TE})$. Equations 3.47 degenerate to 3.48, since the influence from a vortex at infinity is zero. So far the algorithm does not have any advantage over other wake convection algorithms, the only difference being that the doublets are changed into vortices and the convection is carried out using the *quasi-potential* vortex relations, in which the influence of a vortex on itself is zero.

The real novelty of the approach shows up in the unsteady case, where at each time step a doublet is released with length $\Delta s = U_\infty \Delta t$ and with some constant strength μ . Without loss of generality, it can be assumed that the wake is frozen and is convected behind the airfoil with the freestream velocity. At the first time step, one panel is present in the flowfield: therefore it can be split up in two vortices of strength $\Gamma_1 = \mu(t = \Delta t)$ and $\Gamma_2 = -\mu(t = \Delta t)$, $U_\infty \Delta t$ apart. At the second time step, a new doublet of the same length is introduced in the flowfield. The right vertex of the new doublet and the left vertex of the previous one occupy the same location in the plane, therefore they can be thought of as a single vortex placed at that location with strength equal to the algebraic sum of the vortices that occupy the location. Therefore at $t = 2\Delta t$ three vortices are in the flowfield, with strengths respectively (from the right to the left) $\Gamma_1 = \mu(t = \Delta t)$, $\Gamma_2 = -\mu(t = \Delta t) + \mu(t = 2\Delta t)$ and $\Gamma_3 = -\mu(t = 2\Delta t)$. At the n^{th} time step $t = n\Delta t$, there are $n + 1$ vortices in the flowfield. They are

characterized by the following positions and strengths:

$$\left\{ \begin{array}{ll} \Gamma_1 = \mu(t = \Delta t), & x_1 = x_{TE} + U_\infty n \Delta t \\ \Gamma_2 = -\mu(t = \Delta t) + \mu(t = 2\Delta t), & x_2 = x_{TE} + U_\infty (n - 1) \Delta t \\ \dots & \dots \\ \Gamma_j = -\mu(t = (j - 1) \Delta t) + \mu(t = j \Delta t), & x_j = x_{TE} + U_\infty (n - j) \Delta t \\ \dots & \dots \\ \Gamma_n = -\mu(t = (n - 1) \Delta t) + \mu(t = n \Delta t), & x_n = x_{TE} + U_\infty \Delta t \\ \Gamma_{n+1} = -\mu(t = n \Delta t) & x_n = x_{TE} \end{array} \right. \quad (3.49)$$

The system of equations 3.49 can be used also in the case of free wake, with the difference that the velocity with which wake elements convect downstream needs to be computed within the code. In general, at the n^{th} time step, there are $n + 1$ vortices, as shown in figure 3.9, where the passage from doublets (upper figure) to vortices (lower figure) is remarked. One of the vortices occupies the trailing edge position, the others are in the flowfield. The velocity induced at the generic vortex position can be found adding up the contribution from all the singularities in the flowfield. Recalling equation 3.42, the velocity can still be found in the same way, using the vortex relations 3.48.

In section 3.3 the solution strategy has been described: $N + 1$ equations are laid out and allow to find the values of the N unknown singularities on the airfoil and the last doublet shed from the trailing edge. With the new solution strategy proposed, one more unknown is added to

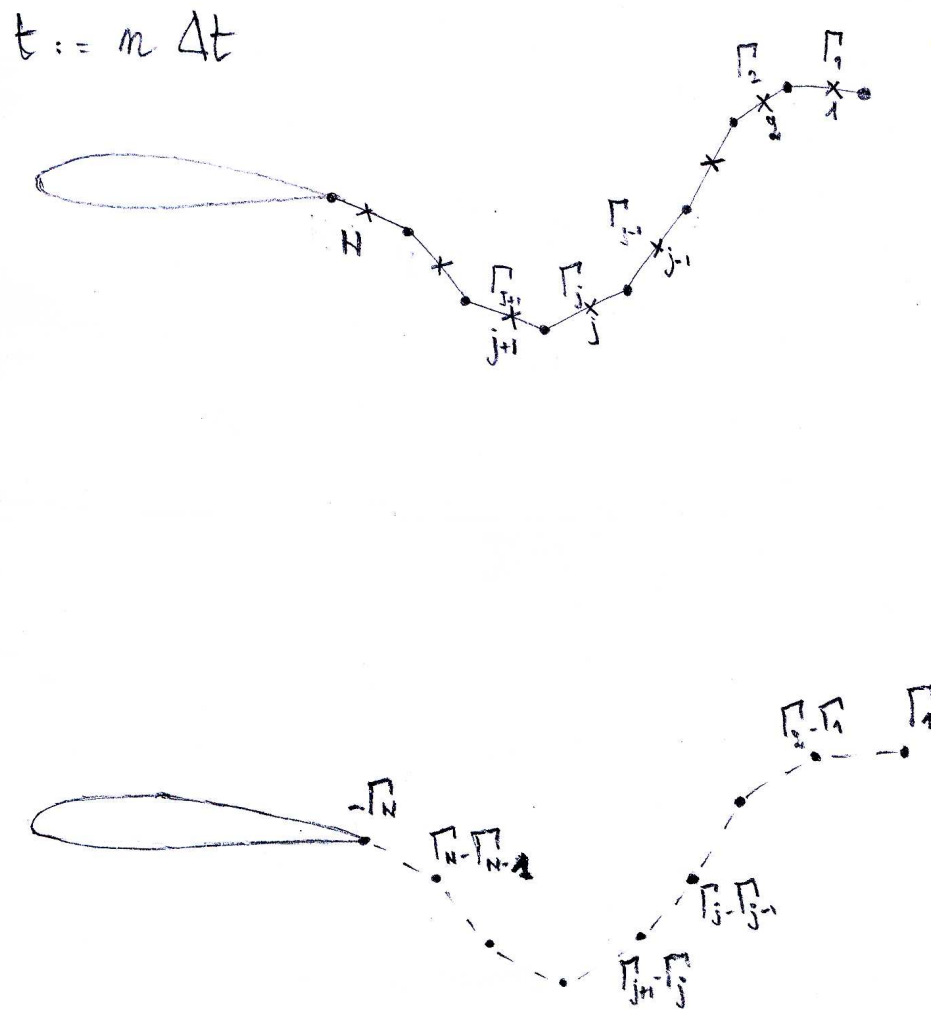


Figure 3.9: Sketch showing the implementation of the panel method applied in the investigation

the system and a new equation needs to be found. This can be done looking at the algorithm 3.49. At time $t_n = n\Delta t$ the 2 Kutta conditions can be expressed in the form:

$$\begin{cases} \Gamma_n &= (\mu_{N_P} - \mu_1) - \Gamma_{n-1} \\ \Gamma_{n+1} &= -(\mu_{N_P} - \mu_1) \end{cases} \quad (3.50)$$

Adding these equations, the $N \times N$ linear system becomes:

$$\begin{bmatrix} a_{1,1} & a_{1,2} & \cdots & a_{1,N_P} & a_{1,N_W} & a_{1,N_W+1} \\ a_{2,1} & a_{2,2} & \cdots & a_{2,N_P} & a_{2,N_W} & a_{2,N_W+1} \\ \vdots & \vdots & \ddots & \ddots & \vdots & \vdots \\ a_{N_P,1} & a_{N_P,2} & \cdots & a_{N_P,N_P} & a_{N_P,N_W} & a_{N_P,N_W+1} \\ -1 & 0 & \cdots & 1 & -1 & 0 \\ 1 & 0 & \cdots & -1 & 0 & -1 \end{bmatrix} \begin{pmatrix} \mu_1 \\ \mu_2 \\ \vdots \\ \mu_{N_P} \\ \Gamma_{N_W} \\ \Gamma_{N_W+1} \end{pmatrix} = \begin{pmatrix} RHS_1 \\ RHS_2 \\ \vdots \\ RHS_{N_P} \\ \Gamma_{t-\Delta t} \\ 0 \end{pmatrix} \quad (3.51)$$

From this point on the solution proceeds as it has been described previously. The difference of this approach from the previous ones lies in the rearrangement of the equations and in a different interpretation of the wake convection process. The importance is that the wake is seen as a continuous sheet, not as a set of vortex points convected independently from each other. In practice, the strength assigned to the vortices is the same as the one found in Jones and Platzer, but the location of the vortices is different. In that case, the vortex strength is concentrated at the centre of the panel. In the present case, the

strength is concentrated where the vortices physically lie. This allows to exploit both the advantages of a continuous wake and the computational efficiency of point vortices. With this new implementation, the programmer does not have to worry whether to convect the wake as a continuous sheet built up of doublets or to convect it as a layer of point vortices; neither has to worry where to place the point vortices in the middle of the panel or at the three-quarter point and to compute the velocity at the quarter chord. Although the derivation has been carried out for constant strength doublets, there is no additional problem if the strength of the doublets varies over the length of the panels. Katz and Plotkin ([12]) show how a doublet of linear varying strength can be split up into two vortices at its vertices. Besides, the same model can be extended to multiple airfoil configurations. It has to be remembered that at the n^{th} time step, $n + 1$ vortices are in the flowfield behind each airfoil, therefore $2(n + 1)$ vortices are in the flowfield in total. Let us consider a configuration that is used in the present thesis as well. An airfoil flapping upstream and a trailing airfoil downstream, with a steady motion. At some instant, the vortices released from the first airfoil will be travelling next to the downstream airfoil. If the wake is continuous and made up of doublets, the induced velocity is computed at the centre of the panel. Therefore, if two vortices are drifted along the body surface, one onto the upper part and the second onto the lower part of the airfoil, the centre of the panel will end up within the contour, violating the condition of no-through flow.

3.6 Vortex Particle Methods

In the introduction of the Vortex Particle Method (VPM) it is our desire to stress the strong link between the latter and the Panel Methods (PM) described in the previous sections. As a matter of fact, a component of VPMs is a Panel Method. In the pattern followed so far in the thesis, a method has been presented with its advantages and drawbacks, always keeping in mind the final goal of the investigation, that is the development of a tool to describe as accurately as possible the flowfield for the design of a flapping wing with aeroelastic constraints. The assumptions made for the development of PM restrict our view to the analysis of (quasi-)potential flows, in which separation cannot be predicted other than at the trailing edge and the viscosity is not taken into account. The VPM allows to remove these assumptions maintaining the computational efficiency of PMs. The idea of using vorticity tracking methods in order to resolve the flowfield comes from the possibility to perform a decomposition of the velocity field into its potential and rotational components, as illustrated in equation 3.52:

$$\mathbf{u}(\mathbf{x}, t) = \frac{1}{2\pi} \int \frac{\omega(\mathbf{x}', t)}{|\mathbf{x} - \mathbf{x}'|^2} \times (\mathbf{x} - \mathbf{x}') d\mathbf{x}' + \nabla\Phi \quad (3.52)$$

Equation 3.52 is the Biot-Savart integral, inversion of the relation $\omega = \nabla \times \mathbf{u}$ between the velocity and the vorticity in the fluid. If the curl of the Navier Stokes equation is taken, and the expression of the vorticity replaces the velocity, the momentum equation can be recast

in terms of the vorticity as the main unknown:

$$\frac{D\omega}{Dt} = \frac{\partial\omega}{\partial t} + \mathbf{u} \cdot \nabla\omega = \nu \nabla^2\omega \quad (3.53)$$

The boundary conditions associated with 3.53 are:

- The wall boundary condition, that is the same one imposed in the panel method. For the case shown above, where the viscosity is not zero, this is the no-slip B.C.
- The conservation of vorticity: Kelvin's theorem states that the vorticity present in the field does not change in time, therefore if it was zero before the motion started, it has to be zero for all the time.

The advantage of 3.53 with respect to the traditional Navier Stokes equations is that there is no nonlinear term in it, and the time derivative is total, therefore the vorticity can be tracked in a Lagrangian manner. Material elements carrying vorticity are confined in a very small space compared with the whole domain that should be discretized if the Eulerian philosophy is adopted. Moreover, the Biot Savart Law fulfills the condition for the velocity to be zero at infinity, therefore the farfield boundary condition needs not to be enforced, as it is automatically satisfied with the formulation employed. The only condition that needs to be enforced is the impermeability of the body surface, that leads to the solution of a system of equations with a

panel-method-like procedure. Equations 3.52 and 3.53 represent the core of the VPM. The basic idea for the solution of the set of equations is then to sample the continuous vorticity field into a finite number of cells with the circulation concentrated on a single point. The circulation in an arbitrary point may then be found by means of the relation:

$$\omega_0 \simeq \omega_0^h = \sum_{i=1}^N \omega_{0i} \delta(\mathbf{x} - \mathbf{x}_i) \quad (3.54)$$

In an inviscid flow the assumption that the circulation around each particle remains constant in time can be made, therefore the only step that needs to be performed is the convection of the particles. The velocity can be found via the Biot Savart relation, in which the approximation of the vorticity by means of particles is introduced:

$$\frac{d\mathbf{x}^h}{dt} = \mathbf{u}^h = \mathbf{K} \star \omega_h \quad (3.55)$$

$$\mathbf{x}_i^h(0) = \mathbf{x}_i \quad (3.56)$$

where $\star$ indicates convolution of the Kernel and the vorticity values. The Kernel $\mathbf{K}$ can assume very high values when the particles approach each other, therefore a procedure to prevent this from happening needs to be addressed. The idea consists in introducing a *mollification* of the kernel in order to have a finite contribution to the velocity when two particles approach each other. As described in section 2.2.5, Krasny devised a way of desingularization through the

introduction of a small parameter ϵ at the denominator of the kernel, such that the velocity would still assume finite values when the positions of two or more particles coincide. A generalization of this approach is formalized in [50] with the introduction of the concept of a *cutoff function* ζ to replace the dirac function δ in 3.54. Introducing a small parameter ϵ and

$$\zeta_\epsilon(\mathbf{x}) = \epsilon^{-2} \zeta\left(\frac{\mathbf{x}}{\epsilon}\right)$$

then the mollified kernel is defined as the convolution of the original kernel with the cutoff function scaled with ϵ :

$$\mathbf{K}_\epsilon = \mathbf{K} \star \omega_h \quad (3.57)$$

The analysis shown in Cottet and Koumoutsakos [50] proves that the method is convergent to the exact solution and that the accuracy depends on the choice of the cutoff function. In general, it can be said that the requirement for the total circulation of a particle to remain constant implies for the integral of the cutoff function to be unitary. It can also be shown, as proved in [50], that the accuracy of the method increases if the cutoff has the same momentum properties of the dirac function. If the circulation is defined as the momentum of *zeroth* order, the linear impulse as the first order momentum and so on, then the approximation is accurate to the *rth* order if the following conditions are satisfied:

$$\begin{aligned}
\int \zeta(\mathbf{x}) d\mathbf{x} &= 1 \\
\int \mathbf{x}^i \zeta(\mathbf{x}) d\mathbf{x} &= 0, \text{ if } |\mathbf{i}| < r - 1 \\
\int |\mathbf{x}|^r |\zeta(\mathbf{x})| d\mathbf{x} &< \infty
\end{aligned} \tag{3.58}$$

Kernels of arbitrary order can be built in order to have higher accuracy. Figure 3.10 shows the comparison of the exact kernel with the 4th and 6th order mollified kernels. As expected, the approximating functions have trends similar to the exact kernel for large values of r . When $r \rightarrow 0$ the mollified kernels tend to zero and the singularity is removed.

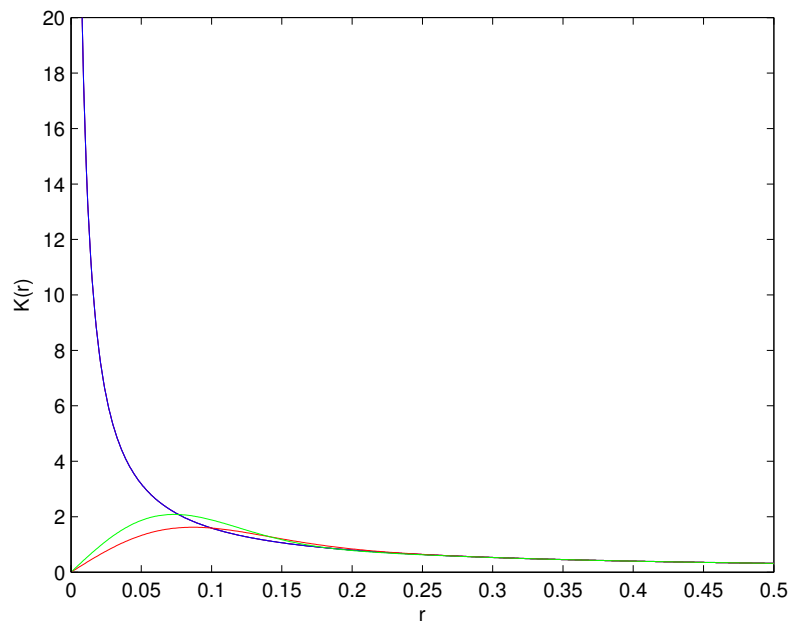


Figure 3.10: Modulus of the exact kernel (blue) compared with the 4th (red) and 6th (green) order mollified kernels for $\epsilon = 0.1$

The solution of equation 3.53 is achieved in two sub-steps:

- *Step 1:* The particles' position is updated integrating Biot-Savart

equation and the vorticity is diffused in the flowfield by means of the particle strength exchange or random walk method. At the end of this substep, a slip velocity exists at the wall;

- *Step 2:* The slip velocity is deleted with the application of a panel method that enforces the no-slip boundary condition onto the body panels.

3.6.1 VPM:Implementation Details

The programming language C++ has been used for the implementation of the Vortex Particle Method, that is seen as a step forward compared to the potential flow methods presented earlier in the chapter.

In the convection and diffusion step, a viscous splitting algorithm is used. First, the particles' position is updated assuming the strength of each particle constant. During this process, the flow is assumed to be inviscid and the particles convect under their own influence and the influence of the vorticity present on the body. Due to the large number of particles in the flowfield, this step requires the use of an accurate time integration scheme. In this thesis a 4th order explicit Runge Kutta scheme has been chosen, in order to have both accuracy and robustness. The use of lower order methods, such as Euler integration methods, would require too small time steps for the solution, therefore an higher order scheme is preferred. The advantage and the drawbacks of this choice will be outlined when the panel method procedure will

be described later in this section. To guarantee that the convection is performed accurately, the choice of the blob function is of primary importance. The velocity needed to perform the convection is given by the discretized form of equation 3.52, where the vorticity approximation is introduced. When the convection occurs in unbounded domains, where no boundary conditions need to be satisfied, the Runge Kutta method can be applied without particular issues. The position of the vortex at the next step is found through the formula:

$$\mathbf{x}_i^{k+1} = \mathbf{x}_i^k + \frac{\Delta t}{6} (\mathbf{f}_1 + 2\mathbf{f}_2 + 2\mathbf{f}_3 + \mathbf{f}_4) \quad (3.59)$$

where the velocities $\mathbf{f}_i, i = 1, \dots, 4$ are computed with the particles at their updated positions in the sub-steps. Lewis [46] shows that an high accuracy can be achieved with a modified Euler scheme, where each sub-step weighs *one* – n^{th} in the whole convection step, where n is the number of the sub-steps used. For instance, when $n = 2$, the convection velocity is the average of two sub-step velocities, the first computed when the particles are in the same positions as they were convected after the previous step, the second with the particles in a $n + \frac{1}{2}$ position. For further details on this integration scheme the reader is referred to [46], in which some examples are given to illustrate the application of the method. In this thesis the 4^{th} order Runge Kutta method is preferred over the modified Euler method because of the number of sub-steps that are needed with the latter in order to achieve the same accuracy. The application of the integration method

becomes more complicated when the boundary condition is satisfied. The vorticity shed is consistent with the impermeability of the body. Therefore, at each sub-step, the vorticity on the body is recomputed in order to guarantee that the Neumann Boundary Condition is fulfilled. To maintain an accurate computation, the vorticity shed off the body panels at the beginning of the subsequent time step is taken as the average of the vorticity found at the sub-steps. The weighting coefficients are the same as the ones used in the Runge Kutta scheme. At each panel location a new vortex is created and shed into the flow-field and then convected with the other vortices. The distance from the body at which the new vortex element is shed into the flowfield is manually entered and it is based on the numerical analysis carried out in the setting up of the parameters. As outlined in [46], the gap by which the new element is displaced from the body is the same order of magnitude as the closest panel length. This problem arose also in the case of trailing edge wake shedding in the panel method and the gained experience leads to the conclusion that a distance equal to the panel length is adequate enough for the shedding. The convection step follows closely the method illustrated in [46] for the convection of vortex elements very close to a bluff body and it will be described here in the most relevant steps. Let us denote the tangential velocity induced on the generic m^{th} panel on the body as $u_{\tau,m} = \frac{1}{2}\gamma_m$, where γ_m is the density of the circulation on the panel. If the vortices were potential, the velocity components (U_{mj}, V_{mj}) at (x_m, y_m) induced by a unitary vortex placed at (x_j, y_j) is given by:

$$\begin{aligned}
U_{mj} &= \frac{1}{2\pi} \left(\frac{y_m - y_j}{r_{mj}^2} \right) \\
V_{mj} &= \frac{1}{2\pi} - \left(\frac{x_m - x_j}{r_{mj}^2} \right)
\end{aligned} \tag{3.60}$$

where r_{mj} denotes the distance between the two points. Then the convection is performed computing the velocity contribution of all the vortex elements in the field with the formulae:

$$u_m = \sum_{j=1}^{N_w} \Gamma U_{mj} + \sum_{j=1}^{N_p} \gamma_j \Delta s_j U_{mj} \tag{3.61}$$

$$v_m = \sum_{j=1}^{N_w} \Gamma V_{mj} + \sum_{j=1}^{N_p} \gamma_j \Delta s_j V_{mj} \tag{3.62}$$

The values Γ of the circulation shed up to the present time step are known. The values that need to be computed are the density of circulation γ over the panels. At this stage the procedure outlined in [46] is closely followed and the attempt is made to adapt the panel method developed in the previous section to the present case. In the panel method developed so far, the singularities representing the body panels and the wake elements are sources and doublets. The formulation of the boundary condition enforcement with this type of singularities leads to the solution of a linear system in which the unknowns are the doublet strengths μ . The advantage of considering constant strength doublets in the panel method is mainly due to the wake shedding: a doublet whose strength is constant over the panel can be seen as a vortex pair placed at the edges of the panel. In the convection step

it is more convenient to handle vortices than doublets, since a vortex is represented by one point and its strength, whereas the doublet needs two points and the strength. The requirement of continuity for the wake shed from the trailing edge is fulfilled automatically with the doublet representation, as outlined in the illustration of the panel method. The procedure applied for the panel method cannot be applied in the present case, because of the number of vortex elements shed at each time step. In the former case the additional unknown μ_W is found introducing the Kutta condition at the trailing edge. In the case of multiple shedding there is no equivalent condition for all the shedding points. Therefore the solution strategy is as follows:

1. Find the strengths of the vortex densities γ on the panels through the enforcement of Neumann boundary condition;
2. Introduce the singularities found on the body into the flowfield and convect them as ordinary vortex elements of strength $\Gamma = \gamma \Delta s$.

The enforcement of the boundary condition yields the values of the vortex strengths on each panel. As shown in the panel method formulation, the boundary condition is enforced in terms of the velocity induced over the panel. If the doublets were used, this step would consist of two sub-steps: first the values of the doublets through the solution of a linear system similar to 3.40 are found, then the computation of the tangential velocity induced on the panel is carried out.

The relation between the induced velocity and the vorticity on the panel is:

$$u_\tau = \frac{\gamma}{2} \quad (3.63)$$

This two step procedure is computationally expensive and much less straightforward to implement, therefore the one presented in [46] is employed. This consists in the choice of vortices, not doublets, as the panel singularities. If the coefficients (U_{mj}, V_{mj}) are considered, the condition for the flow to be parallel to the generic panel can be expressed as:

$$\begin{aligned} \sum_{j=1}^{N_P} K(s_m, s_j) \gamma(s_j) &= \sum_{j=1}^{N_P} (U_{mj} \cos \alpha_m + V_{mj} \sin \alpha_m) \Delta s_j \gamma_j = \\ U_\infty \cos \alpha_m + V_\infty \sin \alpha_m - \sum_{k=1}^{N_W} (U_{mk} \cos \alpha_m + V_{mk} \sin \alpha_m) \Gamma_k \end{aligned} \quad (3.64)$$

The reason for which the condition of flow parallel to the panel replaces the no-through-flow condition is that in the case of constant vorticity over the panel the diagonal terms are zero, that is the normal component of the induced velocity of a vortex on itself is zero. This would make the system ill conditioned, therefore the condition of tangential flow is enforced. In the computation of the left-hand side matrix, it is worth noting that the contribution of the j^{th} panel on the m^{th} one amounts to the contribution of a point vortex placed at (x_j, y_j) with strength $\Gamma_j = \gamma_j \Delta s_j$ with strength on the point (x_m, y_m) . The reason

for this is that the constant strength distribution is used, therefore the strength of the vortex is a factor in the integral. The right hand side of equation 3.64 represents the sum of the free stream tangential velocity and the tangential component of the velocity induced on the m^{th} panel by the k^{th} free vortex in the flowfield. At this stage all the vortices are potential: they are concentrated in one point and their contribution tends to infinity when the distance decreases. Within the framework of the VPM, the free vortices in the flowfield are *mollified* through convolution with a kernel. The vortices placed on the panels remain potential: the reason is that the mollification would lead to part of the vorticity being spread inside the body contour, violating the condition that the body is a streamline. The use of potential vortices over the body creates issues with the convection of the free vortices in proximity of the body. Equations 3.60, representing the velocity induced by a unitary vortex, produce very large values when the free vortex is very close to the panel.

Figure 3.11 shows the velocities induced by a vortex drifted next to a panel lying on the x axis between 0 and 1, with the collocation point at 0.5. The two sets of coloured curves represent two values of the vertical gap between the body and the vortex, respectively 10% (larger gap) and 5% (smaller gap) of the panel length. Decreasing the gap further leads to very large values of the induced velocities at the centre of the panel, therefore the right hand side of 3.64 would be inaccurate. Lewis [46] suggests different techniques for the solution of this issue. The most widely used is the introduction of *subelements*

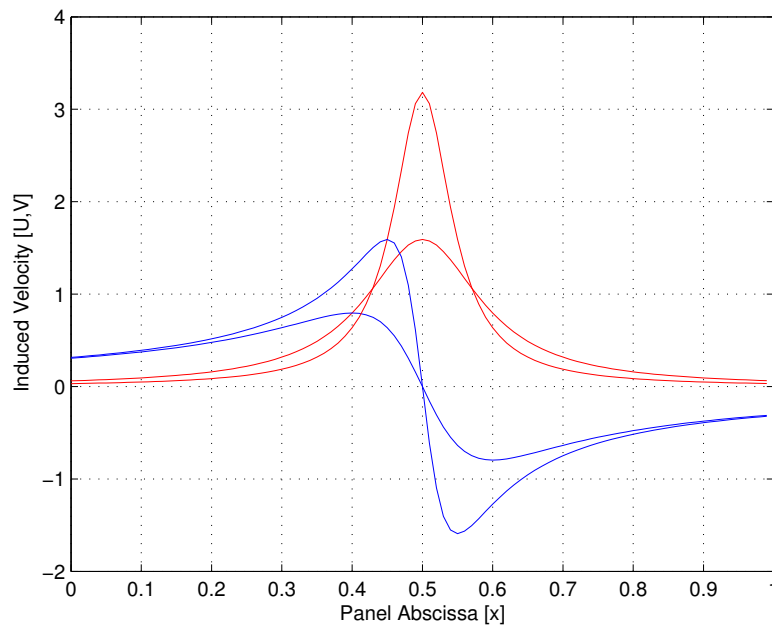


Figure 3.11: Tangential (red) and Normal (blue) Velocities Induced by a Unitary Vortex on a Panel

on the panel. In general the velocity induced by a panel over a close vortex can be thought of as the average of the velocities induced over the same vortex if the vorticity on the panel were spread over the whole panel length:

$$\bar{K} = \frac{1}{\Delta s_n} \int_0^{\Delta s_n} K(s_m, s_n) ds_n \simeq \frac{1}{N} \sum_{i=1}^N N \left\{ \frac{(y_m - y_i) \cos \alpha_m - (x_m - x_i) \sin \alpha_m}{(x_m - x_i)^2 + (y_m - y_i)^2} \right\} \quad (3.65)$$

where the points (x_i, y_i) are defined as:

$$x_i = x_m + \left(i - \frac{1}{2}(1 + N)\right) \frac{\Delta s_n \cos \alpha_n}{N}$$

$$y_i = y_m + (i - \frac{1}{2}(1 + N)) \frac{\Delta s_n \sin \alpha_n}{N} \quad (3.66)$$

With this redistribution, N subelements with equal length and vorticity are introduced. The idea underlying the application of 3.65 is that the contribution of the closest *sub-panels* is weighted with the one coming from all the other elements. The support of the generic subelement is very small, therefore the contribution of this element to the integral in equation 3.65 is bounded. The larger the number of elements, the smoother the function. However, a grid independence study would show that there is no further improvement increasing the number of subelements used beyond a certain level. This is dependent in general on the closeness between the vortex and the panel, therefore on the *singularity level* of the function.

The smoothing in figure 3.12 has been obtained with 100 elements introduced on the panel. The outcome of the computation shows that with this number of subelements the singularity in the middle of the panel is removed. The benefit of the subelements does not concern only the right hand side of the linear system. The difference between the green and the blue curve in figure 3.12 helps understand why: consider a vortex drifted next to the panel represented in the example. If the panel is oriented in the x direction and the bound vorticity is unitary, then U and V are the actual velocities induced on the vortex. The red curve shows that the tangential velocity increases exponentially when the vortex approaches the abscissa of the midpoint. The blue curve, representing the induced velocity normal to

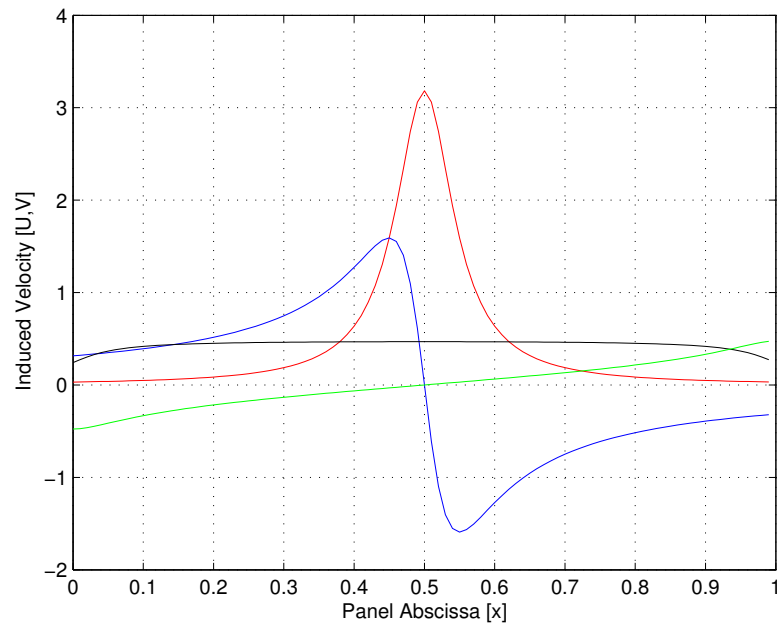


Figure 3.12: The red and blue curves represent induced velocities with no smoothing, the green and black curves represent the same function after subelements have been used

the panel, has a change in sign when the mid-point is crossed. This leads to a *jagged* path of the vortex particle. This inaccuracy becomes disastrous when the gap gets narrower: even though not shown in the figure for scaling reasons, it is straightforward to realize that the induced velocity distribution develops a singularity when the gap tends to zero. Therefore the use of subelements becomes necessary when the vortices travel in very close proximity of the body, such as in the case of vortex clouds.

Other methods that are investigated for smoothing the path of vortex particles are the *Method of Images* and the *Back Diagonal Correction*. The former is not considered in the present implementation, due to the satisfactory results achieved with the subelements. The method of images is widely used in fluid dynamics and consists of placing a mirror image vortex inside the body. This method is mostly used when the whole motion of a single vortex is computed, but it does not have large application in the vortex cloud method. The Back Diagonal Correction consists of enforcing the condition of zero vorticity inside the body, which has not been satisfied so far. It can be done with a small correction on the elements of the matrix of the system and needs very little amount of time for implementation. The idea is that a panel with bound vorticity $\Gamma_m = \gamma_m \Delta s_m$ induces no vorticity inside the body profile. Enforcing it for each panel on the body leads to a condition that needs to be satisfied by the coefficients of the matrix, expressed as:

$$K(s_{opp}, s_m) = -\frac{1}{\Delta s_{opp}} \sum_{n=1, n \neq opp}^M K(s_n, s_m) \Delta s_n \quad (opp = M + 1 - m) \quad (3.67)$$

The implementation of equation 3.67 requires the back diagonal element of each column to be replaced with the sum of the other elements in that column, divided by the length of the panel related to the back diagonal element. The improvement of this technique is demonstrated in [46] with numerical examples. A slightly different technique has been used in this thesis: the element that is replaced is not lying in general on the back diagonal spot. For each column, the largest value is searched and replaced with the sum in equation 3.67. Since the matrix is computed and inverted just once before the time loop starts, the overhead for the back diagonal correction is acceptable. The inversion of the matrix is preferred over the solution of the system at each time step with a Gauss subroutine, because once the inverse is computed and stored, the only operation performed within the time loop is the product of the inverse and the right hand side.

So far the matricial system does not enforce the Kelvin condition like in the panel method. Therefore the conservation of vorticity in the flowfield needs to be satisfied. Since the number of unknowns is equal to the number of equations already, adding one more equation would make the system over-determined. The procedure followed is again taken from Lewis [46]. Since the bound vorticity can be converted into circulation multiplying it for the panel length, the Kelvin equation can

be expressed in the following manner:

$$\sum_{j=1}^{N_P} \gamma_j \Delta s_j + \sum_{k=1}^{N_W} \Gamma_k = 0 \quad (3.68)$$

This equation is added to each of the governing equations enforcing the Neumann boundary condition. The outcome of this modification can be written as follows:

$$\sum_{j=1}^{N_P} (U_{mj} \cos \alpha_m + V_{mj} \sin \alpha_m + 1) \Delta s_j \gamma_j = U_{\infty} \cos \alpha_m + V_{\infty} \sin \alpha_m - \sum_{k=1}^{N_W} (U_{mk} \cos \alpha_m + V_{mk} \sin \alpha_m + 1) \Gamma_k \quad (3.69)$$

The solution of the system 3.69 yields the values of the vorticity on the panels γ , related to the induced tangential velocity through equation 3.63. After the boundary conditions have been enforced, the new vorticity created on the body enters the flowfield and is convected as described above.

The vortex cloud method described in [46] does not perform any kind of particle redistribution. This means that, once shed off the body, the particles are tracked with a Lagrangian scheme, such as in a panel method. Due to the large number of particles in the flowfield after a certain number of time steps, the computation is likely to become inaccurate. A procedure is applied in order to check whether any particle has travelled inside the body contour: if this has happened, the particle is removed from the flowfield but the circulation is still

counted in the conservation of circulation 3.68, that becomes:

$$\sum_{j=1}^{N_P} \gamma_j \Delta s_j + \sum_{k=1}^{N_W} \Gamma_k + \Gamma_{lost} = 0 \quad (3.70)$$

The check is performed following the procedure described in [46], where the penetration of the generic vortex inside the body is detected via the fulfillment of the condition:

$$\begin{cases} \sum_{j=1, j \neq m}^{N_P} r h s_j \Delta s_j < 0.5 \rightarrow \text{Unit vortex is outside the body} \\ \sum_{j=1, j \neq m}^{N_P} r h s_j \Delta s_j > 0.5 \rightarrow \text{Unit vortex is inside the body} \end{cases} \quad (3.71)$$

The element m is the nearest element to the vortex, and in the summation it is omitted for numerical accuracy. The elements detected within the body need to be displaced outwards into the flowfield, or snuffed out and their circulation is taken into account in the Γ_{lost} counter. In this dissertation the elements entering the body contour are snuffed out.

Redistribution Techniques

As explained in section 2.2.5, a fundamental step in the convection and diffusion process of vortex particles is the redistribution of the particles' position. The regularization of the particles' positions is required for higher order schemes, in order to maintain sufficient gaps among particles. In the convection and diffusion, the vortices are not poten-

tial, but have a core radius different from zero. It is important for the particles to be as equispaced as possible in order to diffuse their strength in an accurate manner. Therefore it is necessary, every fixed amount of time steps, to redistribute the particles over a regular grid. The method used for the redistribution is the interpolation of the Lagrangian particles' strength over a new set of particles, initialized over a regular grid. Each particle may be assumed to represent the centroid of a quadrilateral cell with the size closely related to the vortex core radius. The amount of strength exchanged with each particle is proportional to the distance among the particles. As demonstrated in [50], the redistribution scheme works very well when the core of the particles overlap. The scheme that is considered in this dissertation uses polynomial interpolation with the order of the interpolating function chosen depending on the desired accuracy. The functions used are *local*, and the number of elements involved depends on the order of the function. In this thesis the nine-element stencil is used: the Lagrangian element spreads its vorticity to the 9 nearest cells.

The formulae used for the interpolation are not symmetric in the x and y direction over the whole grid. When Lagrangian elements are very close to the body or to the outer grid boundary, the use of symmetric interpolation formulae will cause part of the vorticity to be spread out of the grid. In particular when the element is close to the body, vorticity will be spread inside the body contour, violating the condition of zero vorticity within the body. Figure 3.13 shows the case of particle next to the body contour, where the stencil takes into account that

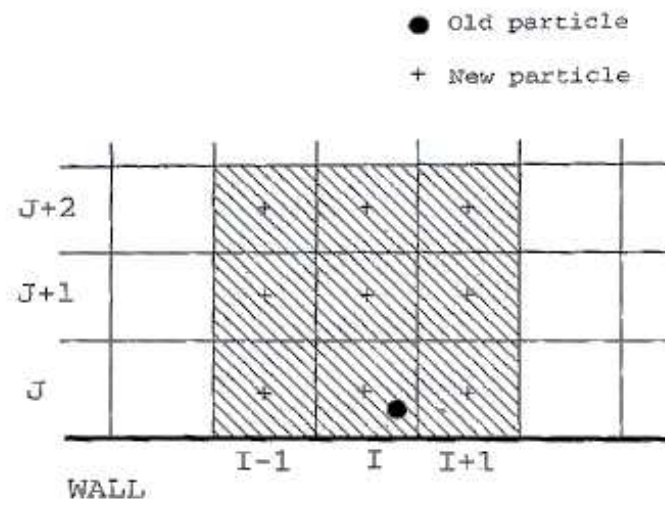


Figure 3.13: The filled dot represents the Lagrangian element, whereas the shaded area denotes the stencil

circulation does not have to spread inside the wall. If i and j denote the tangential and the off-boundary direction, then the symmetric formulae are used when $j = 1, N - 2$, whereas the asymmetric ones are used when $j = 0$ and $j = N - 1$, where 0 denotes the wall cells and $N - 1$ denotes the outer boundary cells. All the interpolation formulae used for the computation of the element redistribution are taken from Cottet and Koumoutsakos [50].

$$\Lambda(x) = \begin{cases} \frac{(1-x)^2(2-x)}{2} & 0 \leq x < \frac{1}{2} \\ \frac{(1-x)(2-x)(3-x)}{6} & \frac{1}{2} \leq x < \frac{3}{2} \\ 0 & x \geq \frac{3}{2} \end{cases} \quad (3.72)$$

$$\Lambda(x) = \begin{cases} 1 - \frac{3}{2}x + \frac{1}{2}x^2 & 0 \leq x < \frac{1}{2} \\ x(2-x) & \frac{1}{2} \leq x < \frac{3}{2} \\ \frac{x(x-1)}{2} & \frac{3}{2} \leq x \leq \frac{5}{2} \end{cases} \quad (3.73)$$

The variable x in the formulae above is the projection of the distance between the particle and the cell centroid on the direction of interpolation. If the off-boundary direction is denoted with ν and the tangential direction with τ , then the local interpolation can be thought of as a 3-by-3 matrix in which the elements on the j^{th} row have the same Λ_ν contribution and the elements on the i^{th} column have the same Λ_τ contribution. The requirement for the redistribution scheme is the conservation of the moments of the vorticity from the $zero^{th}$ to the n^{th} order, where n denotes the order of the scheme. In the present case the Λ scheme is 3^{rd} order accurate, which means that moments up to

the third order are conserved. The interpolating kernel is the product of two one-dimensional kernels:

$$\Lambda(x, y) = \Lambda_\tau(x)\Lambda_\nu(y)$$

3.7 Aeroelasticity of FW

The aim of the thesis is to build a numerical framework in order to be able to perform the aeroelastic analysis of a flapping wing. The motivation for this is the experimental evidence that flexible wings perform better than rigid wings when excited with unsteady motions. It is important to model the aeroelastic effect in the correct manner in order to be able to estimate the forces induced by the flexibility of the wing over the rigid ones. The numerical models that are investigated in the thesis are compared with each other and with others found in the literature in order to assess their advantages and their drawbacks in describing the flow around the wing. The table 3.1 gives an overview of the main strengths and drawbacks of the models described. If the methods are weighed considering only the accuracy and the possibility to model all the phenomena occurring around a flapping wing, then the choice would fall onto the CFD methods. There are and there have been in the last few years attempts to integrate a CFD code within an optimisation framework. In particular it is worth mentioning the work of Lisandrin [113], in which a Reduced Order Model has been

	Strengths	Drawbacks
Theodorsen	Analytical Method Easy to Implement	Not Usable for Generic Configurations
UTIAS	Semi-Analytical Method Improves Theodorsen for Large Oscillations	Fails to Compute High Frequency Oscillations
PM	Numerical Method, Good Accuracy, Quasi Linear	No Stall and Viscosity Modelled
VPM	Very Good Accuracy, Improves Panel Method	Lacks in Accuracy without a grid
CFD	High Order Accuracy Achievable	Complexity Added when Aeroelastic Analysis Required

Table 3.1: Table with pros and cons of using different methods

proposed as an alternative solution to the coupling of structural and aerodynamic solvers. The basis of the method consists in defining a set of structural modes representative of the deformation of the structure. The procedure is carried out in the frequency domain, in which the relation between vector of the aerodynamic forces and the vector containing the state variables is linear, as long as the oscillations are small. In frequency domain, this relation is expressed as:

$$\mathbf{f} = \frac{1}{2}\rho U_{\infty}^2 S \bar{\mathbf{E}} \mathbf{q} \quad (3.74)$$

where $\mathbf{q}$ is the vector containing the generalized coordinates, $\mathbf{f}$ are the generalized forces and $\bar{\mathbf{E}}$ is the matrix of the generalized aerody-

dynamic forces, also known as **GAF**. The GAF is not diagonal in general, showing that when the system is excited along one coordinate only, all the forces will be in general different from zero. If the matrix were diagonal, then the solution of the problem would consist finding the eigenvalues and the eigenvectors of the system, like in structural dynamics. To show this, it is necessary to write the equations of the system *fluid+structure* together, considering the aerodynamic forces as expressed in equation 3.74:

$$s^2 \bar{\mathbf{M}} \tilde{\mathbf{q}} + \bar{\mathbf{K}} \tilde{\mathbf{q}} = \frac{1}{2} \rho U_\infty^2 S \bar{\mathbf{E}} \tilde{\mathbf{q}} \quad (3.75)$$

The left hand side in eq. 3.75 is the structural part of the system, the right hand side is the aerodynamic part. It is important to remark that the aerodynamic forces are not external forces. This is remarked from the fact that they depend upon the generalized coordinates. The external forces, if present in the system, need to be added to the right hand side in order to take them into account. The application of the solution method requires the system to be turned into modal coordinates. The normalization can be done for example multiplying the mass matrix for its inverse. This would change the system into the new one:

$$s^2 \bar{\mathbf{I}} \tilde{\tilde{\mathbf{q}}} + \bar{\mathbf{\Omega}}^2 \tilde{\tilde{\mathbf{q}}} = \frac{1}{2} \rho U_\infty^2 S \bar{\mathbf{E}} \tilde{\tilde{\mathbf{q}}} \quad (3.76)$$

with $\bar{\mathbf{I}}$ being the unitary matrix, $\bar{\mathbf{\Omega}}^2$ is the diagonal matrix whose ele-

ments are the square of the natural frequencies of the structure and $\tilde{\mathbf{q}}$ are the Laplace transforms of the modal coordinates. Bringing the aerodynamic forces to the right hand side yields:

$$\left[s^2 \bar{\mathbf{I}} + \bar{\mathbf{\Omega}}^2 - \frac{1}{2} \rho U_\infty^2 S \bar{\mathbf{E}} \right] \tilde{\mathbf{q}} = 0 \quad (3.77)$$

The last form of the aeroelastic system shows that if the GAF were diagonal, then the techniques of structural dynamics could be used to solve the problem. In particular the equations would be independent, each mode being excited only by the correspondent modal coordinate. Due to the form of the GAF, the equations are coupled and to find the solution requires a different approach. If a reduced order model is used, then a number of representative modes is chosen. Then, the CFD solver is run as many times as many are the modes. Each run yields a column of the GAF matrix, which are the pressure coefficients on the surface of the body projected onto the modal coordinates. When all the runs are performed and the GAF is completed, the eigenvalue analysis is carried out in order to find the frequencies of the aeroelastic system. It is remarkable that the GAF matrix also depends on the complex frequency $s = \zeta + j\omega$. Therefore, the CFD runs should be performed as many times as necessary to cover the frequency range of interest. Lisandrin [113] has used a bell-shaped function to reduce the number of CFD runs to the minimum. In fact, the Fourier Transform of the gaussian function can approximate the one of the dirac distribution, which is constant for all the frequencies. Therefore, changing

the width of the gaussian input fed to the system, the frequency range of interest can be covered. Besides the use of Reduced Order Models, there is no great amount of work carried out to integrate the structural and the CFD solvers into one tool for the aeroelastic analysis. Therefore there does not seem to be much room for the CFD solvers and their integration with structural solvers in the preliminary design phase. Since time is an issue, simpler methods are used. Of course this narrows the field that these methods can investigate, since they are based on more restrictive assumptions than Navier-Stokes or even Euler solvers. In the analysis of fixed wing aircrafts, the oscillations are assumed to be small enough for the linear theory to be applied. Besides, the system of equations 3.77 is valid only in the linear case, because the introduction of the GAF matrix assumes that the forces can be expressed as linear functions of the generalized coordinates. If the problem we are trying to solve is a *stability problem*, the linear approach can be employed. For the linear case, as long as oscillations are small, the potential flow methods will be shown to work very well, under the assumption of inviscid flow. Since the interest is focused in any case on inviscid flow, it seems quite reasonable to develop a method of solution that uses the panel method as the aerodynamic solver. Some non-linearity is added when the wake convection algorithm is *switched on*, since the wake is not bounded to the horizontal plane behind the airfoil. For small oscillations and small frequencies though, the influence of the non-linear wake is limited: when the oscillations are small, the model of Theodorsen seems to hold quite well.

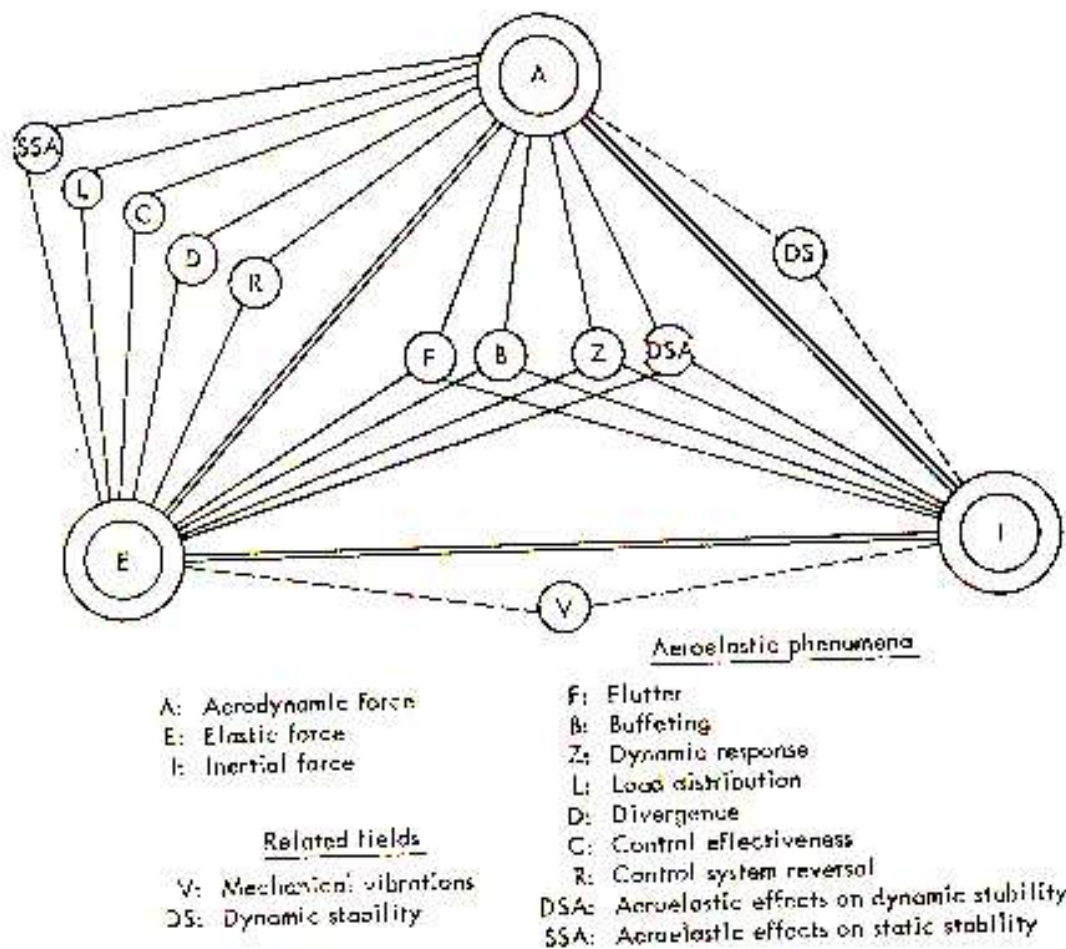


Figure 3.14: Triangle illustrating the phenomena studied when coupling elasticity, dynamics and aerodynamics

Figure 3.14 shows the different phenomena that can be occurring when elasticity, structural dynamics and aerodynamics are coupled together. Our attention is focused on those interactions where the aerodynamics is taken into account. Hence no further talk will be made about the lower edge, where only elasticity and structural dynamics are coupled. The dynamic stability is not considered in this dissertation because we are interested in the interaction of the structure flexibility with the aerodynamic forces. Dynamic stability is part of flight mechanics, where the body is assumed to be rigid. Therefore the phenomena of interest are the divergence on the left edge and the flutter and dynamic response at the centre of the triangle. The divergence can be regarded as a particular case of the dynamic case, with the frequency of oscillation equal to zero. When $s = 0$, the system of equations becomes:

$$\left[\bar{\mathbf{K}} - \frac{1}{2} \rho U_{\infty}^2 S \bar{\mathbf{E}}(s = 0) \right] \tilde{\mathbf{q}} = 0 \quad (3.78)$$

The divergence concerns the static stability of the structure. Equation 3.78 shows that the divergence occurs when the *flow stiffness* overcomes the structural stiffness. This can be visualized in the complex plane, where the real part and the imaginary part of s are plotted for different flow speeds. As long as the system is stable the **poles**, as the eigenvalues of the overall system are called, lie in the negative semi-plane. Increasing the flow speed shifts the poles towards the imaginary axis. When $\zeta = \Re(s)$ crosses the imaginary axis the divergence occurs. When the frequency of oscillation is different from zero, the in-

ertia term has to be added to the system. The two main phenomena that are considered and modelled in the thesis are the flutter and the aeroelastic response of the wing. Theodorsen ([34]) describes the flutter as the instability arising when the flow speed overcomes a critical value, defined as the *flutter speed*. The computation of the flutter speed requires the solution of an eigenvalue problem. If the system in eq. 3.77 is considered again, the eigenvalues are the those values of s for which the determinant of the matrix within square brackets is zero. This is true because the system is homogeneous, therefore it admits a non-trivial solution if and only if the determinant of the matrix is zero. It is important to recall that a n^{th} order differential equation can be turned into a system of n ordinary differential equations of the first order. Let us consider a generic second order system as in 3.76, with the aerodynamic forces equal to zero. Rewriting the system in time domain, we have:

$$\bar{\mathbf{M}}\ddot{\mathbf{q}} + \bar{\mathbf{K}}\mathbf{q} = 0 \quad (3.79)$$

The transformation to a first order system yields:

$$\frac{d}{dt} \begin{Bmatrix} \mathbf{q} \\ \dot{\mathbf{q}} \end{Bmatrix} = \begin{bmatrix} 0 & \mathbf{I} \\ -\mathbf{M}^{-1}\mathbf{K} & 0 \end{bmatrix} \begin{Bmatrix} \mathbf{q} \\ \dot{\mathbf{q}} \end{Bmatrix} \quad (3.80)$$

The eigenvalue problem for such a system can be written as:

$$\bar{\mathbf{A}}\mathbf{v}^n = \mu_n \mathbf{v}^n \quad (3.81)$$

with the eigenvectors decomposed in two parts:

$$\mathbf{v}^n = \begin{Bmatrix} \mathbf{v}_1^n \\ \mathbf{v}_2^n \end{Bmatrix} \quad (3.82)$$

Let us consider the same problem for the second order system:

$$\begin{aligned} \ddot{\mathbf{q}} + \bar{\mathbf{M}}^{-1}\bar{\mathbf{K}}\mathbf{q} &= 0 \\ \bar{\mathbf{M}}^{-1}\bar{\mathbf{K}}\mathbf{Z}^n &= \lambda_n \mathbf{Z}^n \end{aligned} \quad (3.83)$$

If we consider that the matrix $\mathbf{A}$ in 3.81 is the same as given in 3.80, it is readily found:

$$\begin{aligned} \mathbf{v}_2^n &= \mu_n \mathbf{v}_1^n \\ -\bar{\mathbf{M}}^{-1}\bar{\mathbf{K}}\mathbf{v}_1^n &= \mu_n \mathbf{v}_2^n \end{aligned} \quad (3.84)$$

Combining both equations 3.84 yields:

$$-\bar{\mathbf{M}}^{-1}\bar{\mathbf{K}}\mathbf{v}_1^n = \mu_n^2 \mathbf{v}_1^n \quad (3.85)$$

Comparing 3.85 and the second of 3.83, it can be obtained:

$$-\mu_n^2 = \lambda_n \quad (3.86)$$

Since $\lambda_n > 0$, it is concluded that

$$\mu_n = \pm j\sqrt{\lambda_n} \quad (3.87)$$

This explains why in the first order system the eigenvalues are complex conjugates, whereas in the second order system they are real. The same relation found for the eigenvalues applies to the eigenvectors:

$$\mathbf{v}^n = \begin{Bmatrix} \mathbf{v}_1^n \\ \mathbf{v}_2^n \end{Bmatrix} = \begin{Bmatrix} \mathbf{Z}^n \\ \pm\sqrt{\lambda_n}\mathbf{Z}^n \end{Bmatrix} \quad (3.88)$$

This procedure shows the relation between the eigenvalues of the second order and first order system. Therefore, to find the values for which the determinant of 3.77 is the same as to find the eigenvalues of the first order system. The dependence of the GAF upon the complex frequency s is not reducible to a sum of powers of s . To understand the dependence of the GAF on s , it has to be recalled that the wake introduces a delay in time, that is represented by a pole in the Laplace domain. The approximation of the GAF is given in the form:

$$\bar{\mathbf{E}} = \bar{\mathbf{A}}_0 + \bar{\mathbf{A}}_1 p + \bar{\mathbf{A}}_2 p^2 + (p\bar{\mathbf{I}} - \bar{\mathbf{P}})^{-1} p \bar{\mathbf{P}} \quad (3.89)$$

Bringing the aerodynamic terms to the left hand side, it finally yields:

$$\left\{ [\bar{\mathbf{I}} - \frac{1}{2}\rho_\infty l \bar{\mathbf{A}}_2] s^2 - q_D \frac{l}{U_\infty} \bar{\mathbf{A}}_1 s + (\bar{\boldsymbol{\Omega}}^2 - q_D \bar{\mathbf{A}}_0 - q_D \bar{\mathbf{R}}) \right\} \tilde{\mathbf{q}} - \tilde{\mathbf{r}} = 0 \quad (3.90)$$

with $\tilde{\mathbf{r}} = q_D (\frac{sl}{U_\infty} \bar{\mathbf{I}} - \bar{\mathbf{P}})^{-1} \bar{\mathbf{P}} \bar{\mathbf{R}} \tilde{\mathbf{q}}$, the added state. The dynamic of $\mathbf{r}$ is described by the equation obtained multiplying the definition above by the matrix $[p\bar{\mathbf{I}} - \bar{\mathbf{P}}]$. If the system is anti-transformed to time domain, the final expression is:

$$\begin{cases} \frac{l}{U_\infty} \dot{\mathbf{r}} = \bar{\mathbf{P}} \mathbf{r} + q_D \bar{\mathbf{P}} \bar{\mathbf{R}} \mathbf{q} \\ \bar{\mathbf{M}}_C \ddot{\mathbf{q}} + \bar{\mathbf{D}}_C \dot{\mathbf{q}} + \bar{\mathbf{K}}_C \mathbf{q} - \mathbf{r} = 0 \end{cases} \quad (3.91)$$

where $\mathbf{M}_C$, $\mathbf{D}_C$ and $\mathbf{K}_C$ denote the total mass, damping and stiffness matrices, as it can be obtained from equation 3.90. If N is the initial number of unknowns, the procedure outlined has increased the number of unknowns to $3N$. The final form of the system is:

$$\dot{\mathbf{x}} = \bar{\mathbf{A}} \mathbf{x} \quad (3.92)$$

with

$$\mathbf{x} = \begin{Bmatrix} \mathbf{q} \\ \dot{\mathbf{q}} \\ \mathbf{r} \end{Bmatrix}$$

and

$$\mathbf{A} = \begin{bmatrix} 0 & \bar{\mathbf{I}} & 0 \\ -\bar{\mathbf{M}}_C^{-1}\bar{\mathbf{K}}_C & -\bar{\mathbf{M}}_C^{-1}\bar{\mathbf{D}}_C & \bar{\mathbf{M}}_C^{-1} \\ q_D \frac{U_\infty}{l} \bar{\mathbf{P}}\bar{\mathbf{R}} & 0 & \frac{U_\infty}{l} \bar{\mathbf{I}} \end{bmatrix}$$

The finite state aerodynamic approximation introduces a number of added states to the system, that take into account the wake shed. The number of states added depends upon the wake taken into account. The same thing can be said when approximating the Theodorsen function. The Pade' approximation of $C(k)$ given in 3.3 can be written as:

$$C(k) \simeq \frac{1}{2} \frac{(p + 0.135)(p + 0.651)}{(p + 0.0965)(p + 0.4555)}$$

The eigenvalues of the system in 3.92 can be obtained with the usual procedure. These eigenvalues are plotted on the complex plane for different values of freestream velocity. This method is not considered in the thesis because we want to take all the wake into account. It has been used in the discussion in order to show the dependence of the GAF matrix from the complex frequency. The method that is actually used in this context is based on the computation of the response of the system. This requires the use of a numerical integration procedure for the solution of systems of ordinary differential equations.

A two dimensional wing section model employed for aeroelastic coupling between the structure and the airflow is shown in figure 3.15. The model has 2 degrees of freedom, i.e. vertical displacement and rotation around the elastic axis. The linear and rotational springs

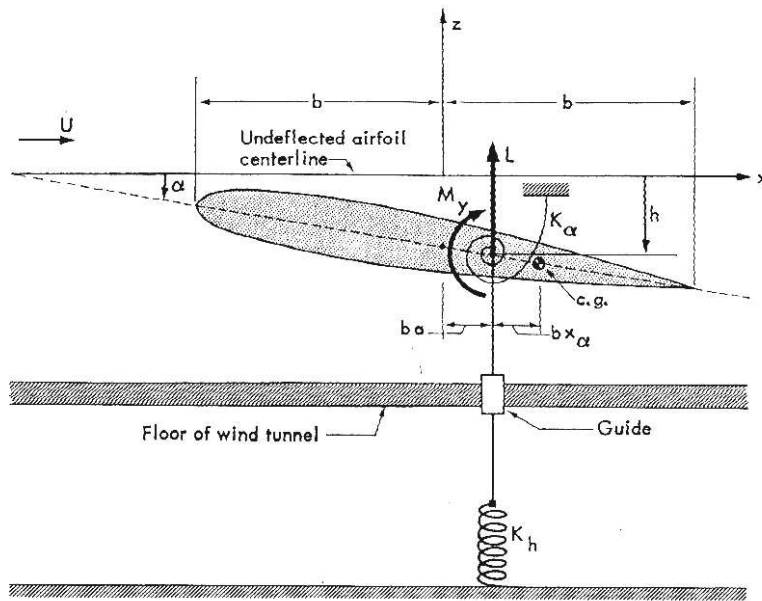


Figure 3.15: Semi-rigid airfoil oscillating in pitch/plunge

represent the stiffness of the wing, whereas the mass properties are given by the mass, the static and inertial moment about the center of rotation. The equations of motions are obtained by writing the kinetic and potential energy of the model. The horizontal (u) and vertical (w) displacements of the airfoil can be written as:

$$\begin{aligned} u &= -r \cos \alpha \\ w &= -h - r \sin \alpha \end{aligned} \quad (3.93)$$

The kinetic energy is obtained deriving the expression of the displacement with respect to time:

$$T = \frac{1}{2} \int_{chord} (\dot{u}^2 + \dot{w}^2) dm$$

$$= \frac{1}{2}m\dot{h}^2 + \frac{1}{2}I_\alpha\dot{\alpha}^2 + S_\alpha\dot{\alpha}\dot{h}\cos\alpha \quad (3.94)$$

The strain energy of the airfoil is given by:

$$U = \frac{1}{2}k_h h^2 + \frac{1}{2}k_\alpha \alpha^2 \quad (3.95)$$

Finally the equations of motions for the two-degree-of-freedom airfoil can be expressed as:

$$\begin{bmatrix} m & x_\alpha m \\ x_\alpha m & I_\alpha \end{bmatrix} + \begin{bmatrix} \omega_h^2 & 0 \\ 0 & \omega_\alpha \end{bmatrix} = \frac{1}{2}\rho U_\infty^2 c \begin{Bmatrix} c_L \\ c_{M\alpha} \end{Bmatrix} \quad (3.96)$$

In the model developed by Theodorsen the analytical expression of the right hand side is available, therefore the problem can be solved by bringing the right hand side to the left and finding the eigenvalues of the system. This is accomplished by a finite state approximation or by other methods such as p - k . When the forces are not available analytically, a numerical method is applied in order to integrate eq. 3.96. The solution yields the vector $\mathbf{x}$ of the unknowns. The above equations can be non-dimensionalized [43] by dividing the first row by $mb\omega_\alpha^2$ and the second by $mb^2\omega_\alpha^2$. The outcome is:

$$\begin{bmatrix} 1 & \xi_\alpha \\ \xi_\alpha & r_\alpha^2 \end{bmatrix} \begin{Bmatrix} \ddot{h} \\ \ddot{\alpha} \end{Bmatrix} + \Omega^2 \begin{bmatrix} 1 & 0 \\ 0 & \frac{r_\alpha^2}{\Omega^2} \end{bmatrix} \begin{Bmatrix} h \\ \alpha \end{Bmatrix} = \frac{\hat{U}^2}{\pi\mu_{AD}} \begin{Bmatrix} -C_L \\ 2C_{M\alpha} \end{Bmatrix} \quad (3.97)$$

where $\hat{U}$ is the non-dimensional velocity, Ω is the ratio between the

plunge and pitch uncoupled frequencies, μ_{AD} is the ratio between the airfoil's and the surrounding airflow's mass, ξ_α is the distance of the center of gravity from the elastic center and r_α is the radius of gyration, both expressed in semi-chord units. It is important to remark that the reference point to compute the moment is the elastic center, at which the springs are concentrated [115, 116]. This allows a direct comparison of results with Theodorsen method, which takes the elastic center as the origin of the system, the rotation point and the point about which the moment is computed.

In order to have a first-order differential equation system, Equations 3.97 are recast in normal form, with the solution found by the traditional techniques of structural dynamics. The scheme used for the solution is based on Newmark method ([105]). In linear aeroelasticity, equations 3.97 can be used to study the flutter boundary of the airfoil. On the other hand, when the motion of a wing in plunge and pitch is known a priori, the response can be determined and plotted for several values of freestream velocity. If the response is damped out, the system is stable; otherwise unstable. Time domain simulations normally require a long time before reaching any instability, because the amplification can be very small. It would be very computational time consuming to find the flutter speed of an airfoil in time domain. Therefore the approach adopted in this paper is to take Theodorsen method as a first approximation of the flutter speed. The panel method can be then applied to refine the calculation in a narrowed velocity range, taking into account that the flutter speed should be below Theodorsen

results.

Once the response of a fixed wing free to move in pitch and plunge has been studied, the aim of the present investigation was to assess whether advantage can be gained from considering a flexible flapping wing. If the same configuration shown in figure 3.15 is considered and the system is undergoing oscillations in pitch and plunge, the equations describing the dynamics are different from 3.97, for the inertial forces need to be added on the right-hand-side. The total displacement of the wing can be thought of as an elastic one on top of the rigid one, that is prescribed in advance. The equations then write as:

$$\begin{bmatrix} 1 & \xi_\alpha \\ \xi_\alpha & r_\alpha^2 \end{bmatrix} \begin{Bmatrix} \ddot{h}_{el} \\ \ddot{\alpha}_{el} \end{Bmatrix} + \Omega^2 \begin{bmatrix} 1 & 0 \\ 0 & \frac{r_\alpha^2}{\Omega^2} \end{bmatrix} \begin{Bmatrix} h_{el} \\ \alpha_{el} \end{Bmatrix} = \frac{\hat{U}^2}{\pi \mu_{AD}} \begin{Bmatrix} -C_L \\ 2C_{M\alpha} \end{Bmatrix} - \begin{bmatrix} 1 & \xi_\alpha \\ \xi_\alpha & r_\alpha^2 \end{bmatrix} \begin{Bmatrix} \ddot{h}_0 \\ \ddot{\alpha}_0 \end{Bmatrix} \quad (3.98)$$

In equations 3.98 the right-hand-side contains the aerodynamic forces, that are computed considering the sum of the elastic and the rigid body motions, and the inertial forces produced by the rigid body motion of the flapping wing only. The aerodynamic solver receives as input the total displacement of the wing at each time step, and outputs the forces acting on the wing. The structural system receives the forces from the aerodynamics, couples them with the mass and stiffness properties of the wing and with the inertial forces produced by the motion, and outputs the elastic deformations.

In a purely structural system, to excite the wing in plunge would

yield a response only along this degree of freedom. In the present case this would occur only if $\xi_\alpha = 0$, therefore when elastic center and center of gravity lie at the same position. Due to the presence of the airflow, when a flapping wing is excited in plunge, it will respond with an elastic bending coupled with a twisting deformation. The present investigation is made to assess whether the elastic deformation, due to aeroelastic effect, would produce additional forces over one cycle, i.e. greater thrust and lift coefficients.

Such aeroelastic beneficial effect provides an advantage for the design of a flapping wing Micro Air Vehicle, because it would be feasible to make the flapping motion simpler and the wing lighter. Although one may attempt to increase the amplitude of the excitation to obtain a greater response, it must be kept in mind the the model employed has limitations due to its assumptions. The main assumption is for the flow to remain attached to the airfoil's surface up to the trailing edge. This limits the amplitude of the oscillations to small values. Although the panel method can compute the real shape of the wake released from the airfoil, it is noted that the wake is shed from the trailing edge and is infinitely thin. Therefore no stall can be predicted with the current model. This limits the amplitude of the oscillations for calculating the aeroelastic response to small values.

Chapter 4

Results and Discussions

This chapter is devoted to showing the results of the tests carried out with the models described in the previous chapter. Part of the results can also be found in [117, 118, 119]. The analysis of the results starts with the comparison of the unsteady aerodynamics predicted with the potential flow methods, in particular with Theodorsen and the Deformable Wavy Wake model. Theodorsen remains the benchmark of every potential flow model, since it is known to be the exact solution for small oscillations. The chapter will then explain the reasons that led to the development of the panel method as a natural extension to the Deformable Wavy Wake model, based on the analysis of the results of both the methods. The improvement achieved with the panel method compared to the other potential flow models associated with the minimum overhead in terms of computational time suggest this to be the right way to proceed towards the development of a computational tool for the aeroelastic analysis. Different configurations are proposed, with single and multiple airfoils flapping with several frequencies and am-

plitudes. Quite a large portion of the chapter is devoted to the analysis of the results obtained with the development of the Vortex Particle Method. They have been introduced in the thesis although further work needs to be done to improve the capabilities of the code. The results presented in here show the improvements in terms of wake shedding and convection when adopting a Lagrangian point of view and the potential of using such a method for time-marching simulations. The last part of the chapter shows the results obtained with the integration of the panel method within an aeroelastic stability and response analysis tool. These results are compared with the data available in literature, showing a reasonable agreement.

4.1 Deformable Wavy Wake

The section is dedicated to the results obtained with the method developed by Fairgrieve and DeLaurier ([40]) and re-implemented in order to be used for comparison with other methods. The method makes an attempt to bridge the gap between Theodorsen and the Panel Method developed in the next section. The improvement obtained with respect to Theodorsen will be shown step by step, first considering the flat wake model, then adding up complexity with the frozen wake model and at last presenting the model with the free wake convection. Figures 4.1 and 4.2 show an important phenomenon that occurs during an impulsively started motion of a wing: *the Wagner Effect*. It is well known that the lift does not reach its steady value instantaneously

during the flapping motion, as described in [30]. The reason for this is the vortex released at the beginning of the motion. The presence of the starting vortex prevents the wing from developing an instantaneous steady lift, which is approached gradually as the vortex travels downstream. The effect of the starting vortex is much more pronounced in insect-like flapping wings, as the wing stops and inverts the motion after having travelled only a few chords. Figure 4.2 shows the ratio between the instantaneous lift and the steady lift, that for a flat plate is $2\pi\alpha$. The result is compared with [40], pp.53-54. The agreement between the results is no surprise, since the models used for the analysis are the same. The comparison is carried out in order to develop a reliable model that is going to be compared with different models in the next paragraphs.

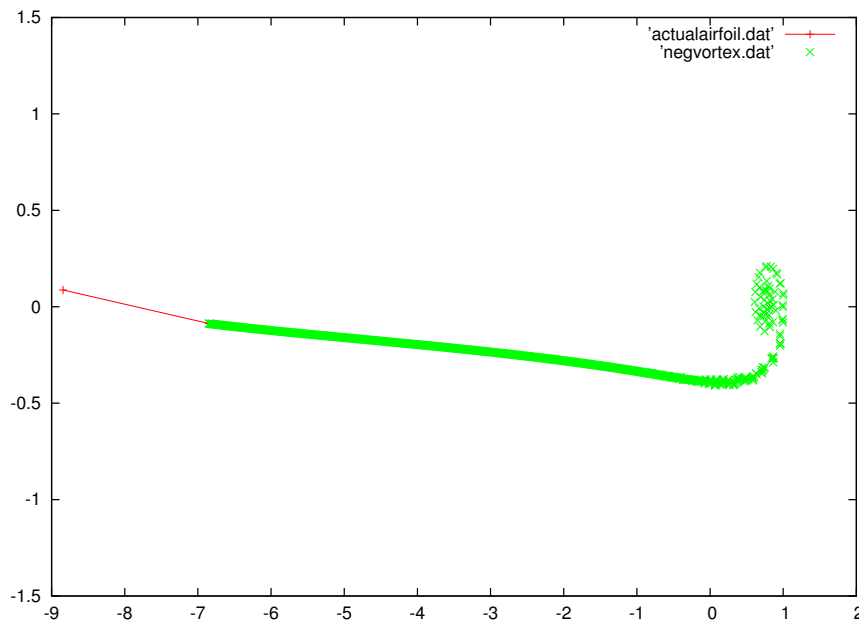


Figure 4.1: Roll up of the wake behind a flat plate with 5 degrees angle of attack

The results in the present paragraph aim at validating the Deformable Wavy Wake model against Theodorsen. In particular, examples are used

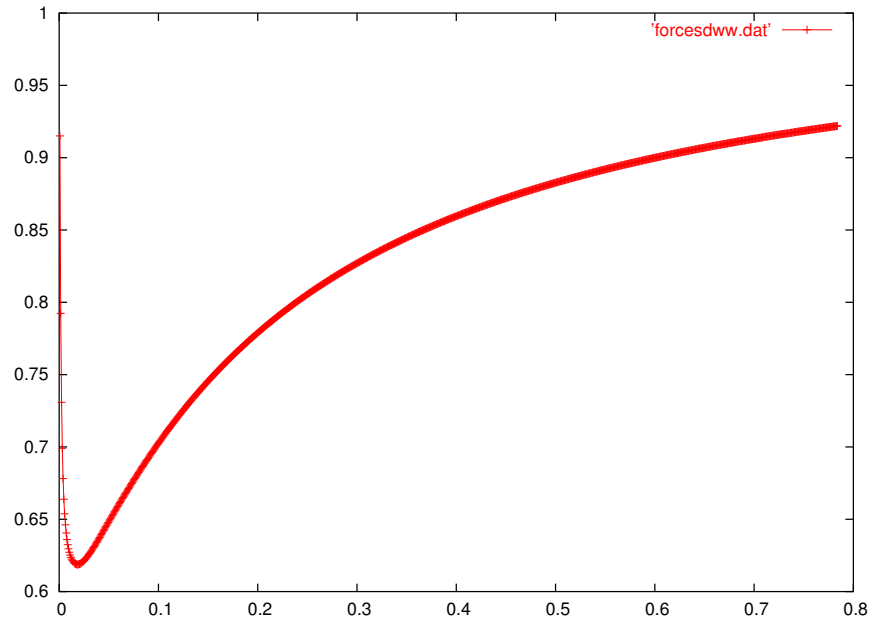


Figure 4.2: Build up of the lift during an impulsively started motion

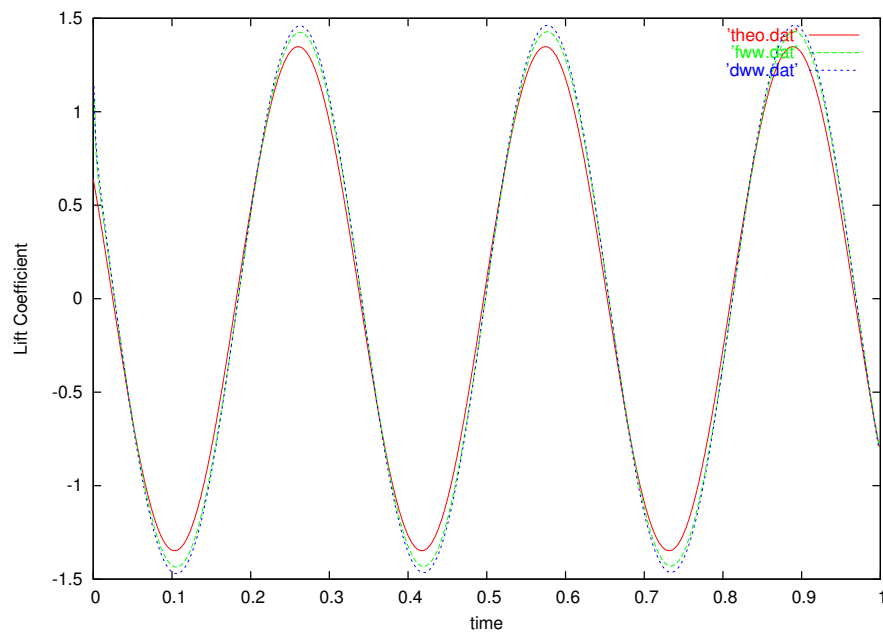


Figure 4.3: $\omega = 20\text{rad/s}$, $h = 0.1$

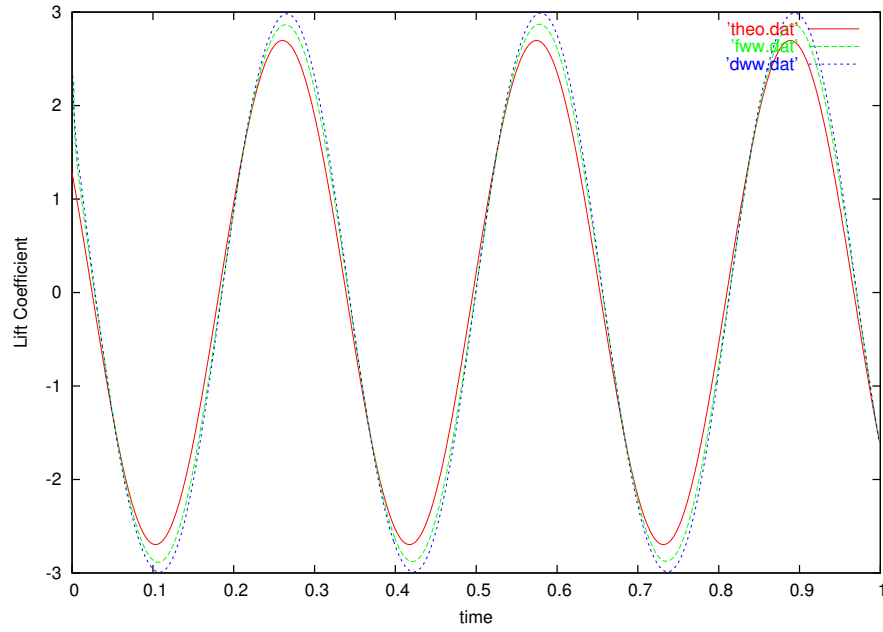


Figure 4.4: $\omega = 20\text{rad/s}$, $h = 0.2$

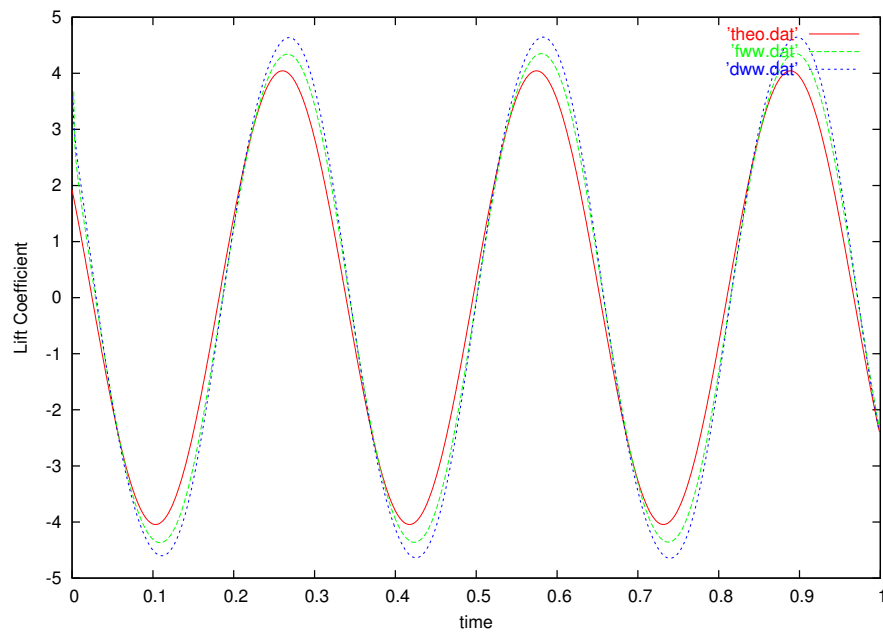


Figure 4.5: $\omega = 20\text{rad/s}$, $h = 0.3$

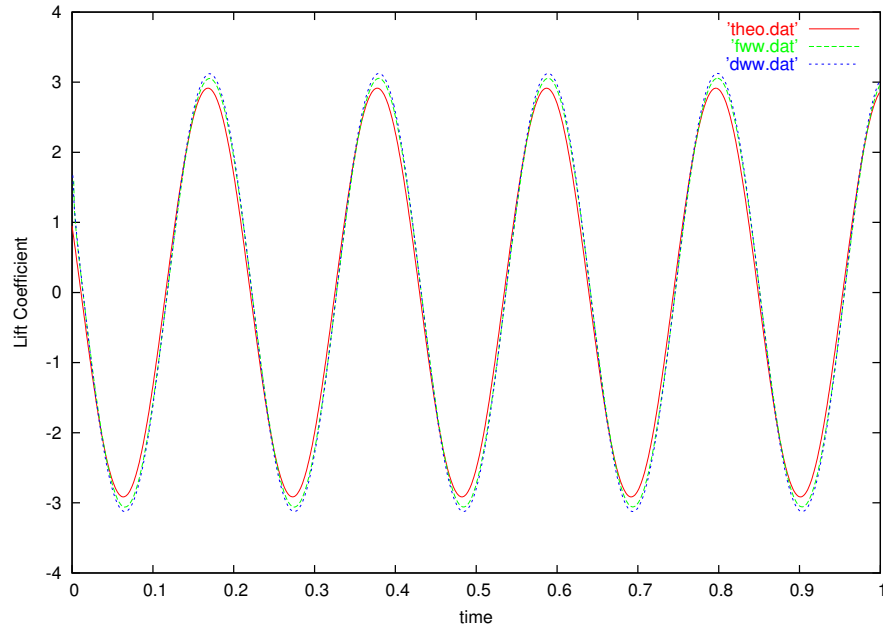


Figure 4.6: $\omega = 30\text{rad/s}$, $h = 0.1$

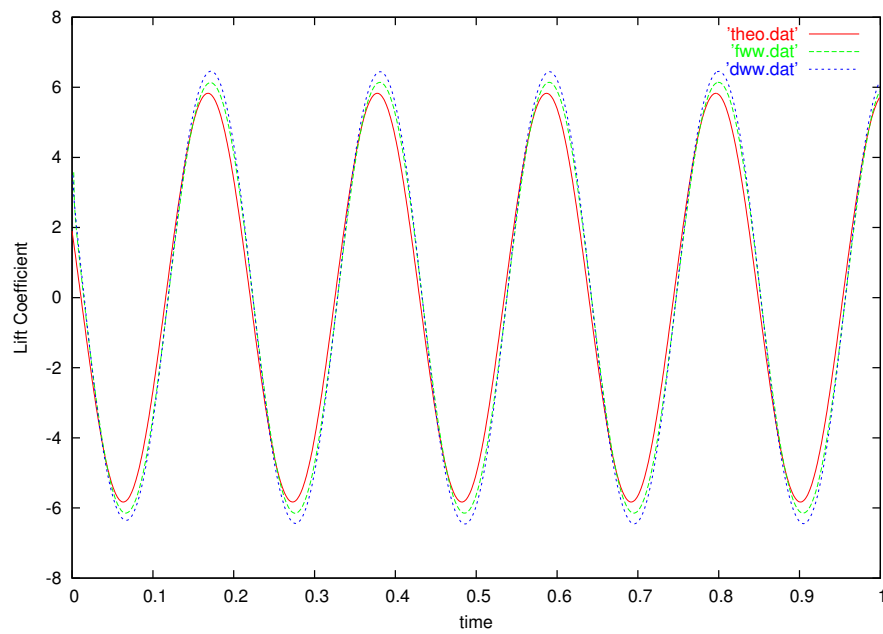


Figure 4.7: $\omega = 30\text{rad/s}$, $h = 0.2$

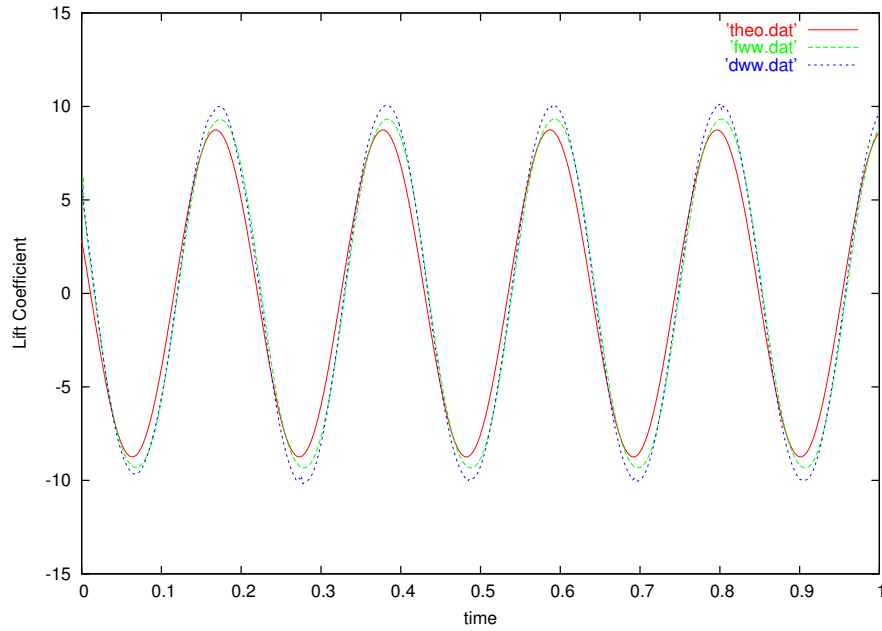


Figure 4.8: $\omega = 30\text{rad/s}$, $h = 0.3$

to demonstrate the reliability of the DWW model under small oscillation assumption. Figures 4.5-4.8 show the comparison among the three models used for the flat plate analysis. The Frozen Wavy Wake model (FWW) is the same as the DWW, with the difference that the wake elements remain frozen in the positions where they are released from the trailing edge. The FWW is in between Theodorsen and DWW. The figures show a gap between Theodorsen and FWW: Fairgrieve and DeLaurier have shown that this gap is mainly due to the numerical model employed, with the out-of-plane displacement of the wake having little effect on it. Therefore the FWW model can be regarded as equivalent to Theodorsen. It is worth remarking that a gap arises also between FWW and DWW, indicating that the deformable wake does have an effect on the forces produced by the flat plate. Besides, the greater the frequency and the amplitude, the greater the gap between FWW and DWW. For the combination of large frequency and large

amplitude, a phase difference appears as well, indicating that the function of Theodorsen does not completely describe the wake influence on the plate. From figures 4.5-4.8 it can also be seen that the gap between FWW and DWW increases with the frequency.

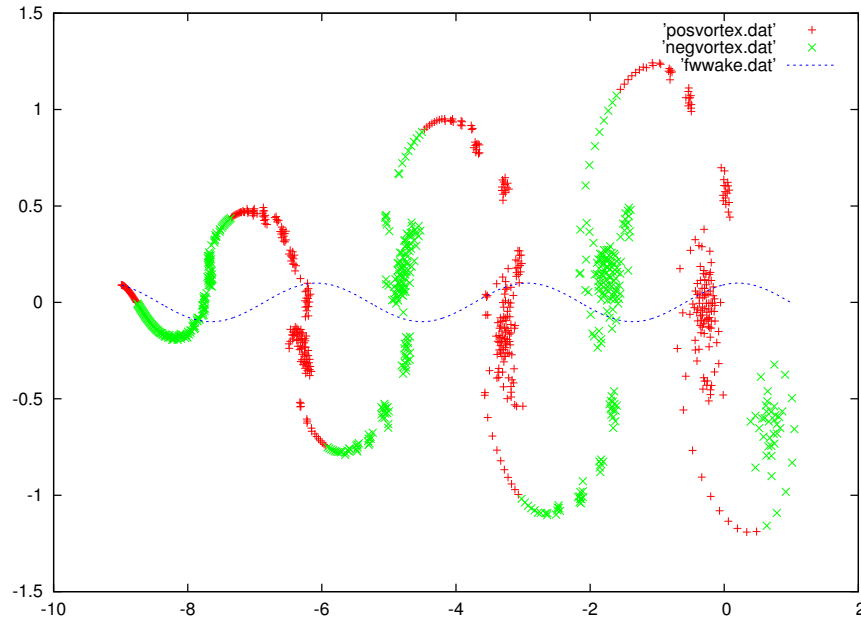
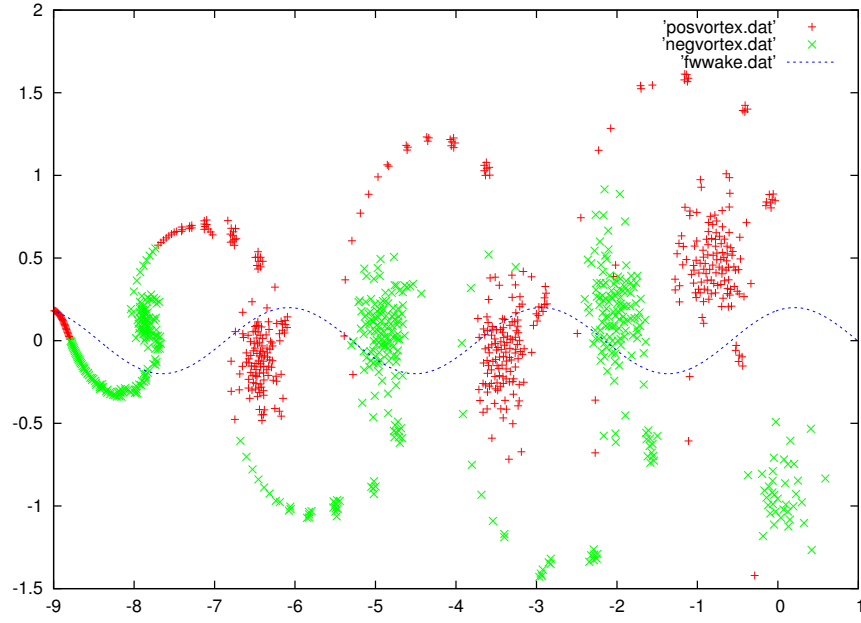
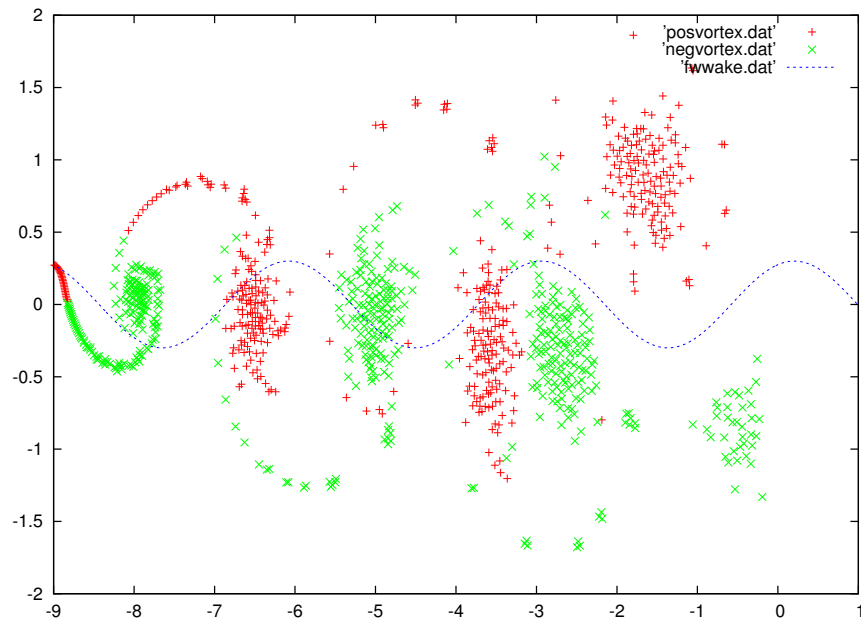
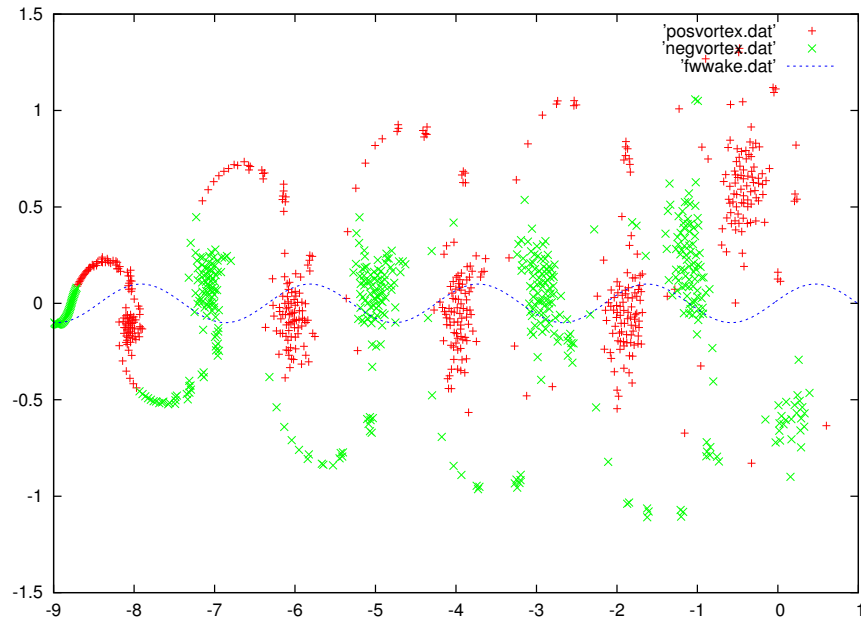
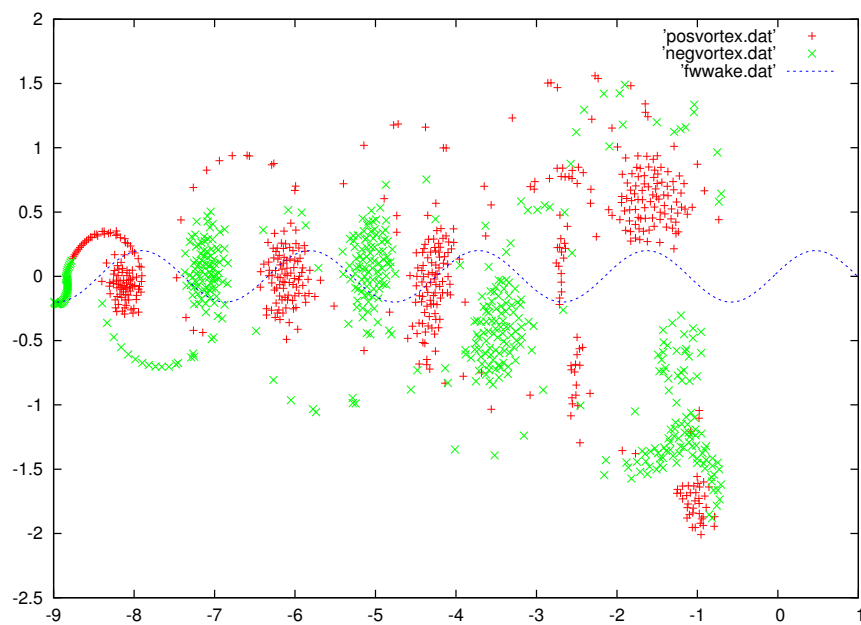


Figure 4.9: $\omega = 20 \text{ rad/s}$, $h = 0.1$

Figures 4.11-4.14 show the wake patterns for the same cases for which the forces have been discussed. As it can be observed, the wake is far from being flat: this can be indicated as the main reason of the discrepancy between Theodorsen and DWW. The assumption of frozen wake is not correct either: the figures show that the error committed when considering the wake frozen cannot be neglected. This reflects upon the forces computed with the two methods. Therefore the FWW method is not considered in the subsequent analysis, as it can be regarded as a modified Theodorsen method. If the wake displacement from the centreline is not enough to justify the inadequacy of Theodorsen for large amplitude motions, the wake roll up is. From now on the DWW will be

Figure 4.10: $\omega = 20\text{rad/s}$, $h = 0.2$ Figure 4.11: $\omega = 20\text{rad/s}$, $h = 0.3$

Figure 4.12: $\omega = 30\text{rad/s}$, $h = 0.1$ Figure 4.13: $\omega = 30\text{rad/s}$, $h = 0.2$

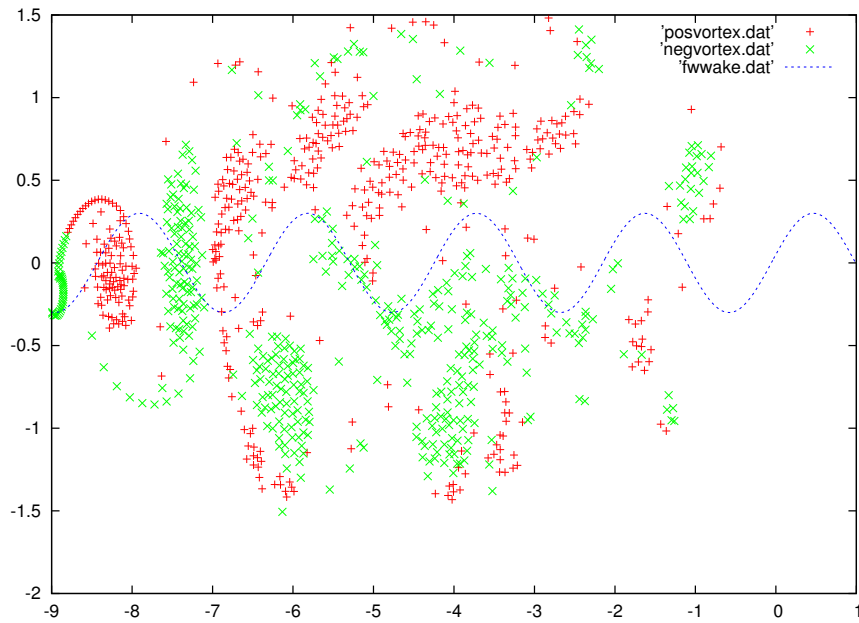


Figure 4.14: $\omega = 30\text{rad/s}$, $h = 0.3$

considered as a more accurate representation of the unsteady forces acting on a flat plate under the assumption of inviscid flow. Another interesting aspect that can be observed is the asymmetry of the wake for the large frequency oscillations. This had already been pointed out by Jones and Platzer [8], who observed the asymmetry for the highest frequency computed. Figure 4.14 highlights the asymmetry plotting on the same graph both the deformable and frozen wakes. In the low frequency cases, the vortex elements seem to be lumped on the x axis. For the larger frequencies, they seem to have an outward drift velocity that triggers the asymmetric path. This can be regarded as the main reason for the rising of a phase delay in the unsteady forces produced with the DWW compared to Theodorsen.

In conclusion, it can be said that the advantage of the DWW method compared to the linear theory lies in its capability of describing the

wake deformation when the oscillations are large. The results show that neither the wake nor the forces can be represented accurately if linear methods are employed. It can be argued that the potential flow theory is not able to represent the forces accurately and that a Navier Stokes approach would be more suitable for this purpose. The missing part in the linear theory is not the accuracy, but the impossibility to represent correctly the main phenomena occurring in the flowfield. Therefore the use of a more complex method, which releases some assumptions made within Theodorsen, is a clear step forward towards the correct representation of the unsteady aerodynamics.

4.2 Single Airfoil Flapping

The results presented in this section have been partly presented in Liani and al. [119]. They have been divided in two parts: single airfoil flapping and multiple airfoil flapping. In the first part an accurate analysis of the unsteady panel method is performed. The advantages compared to the linear theory are outlined, together with the aspects in which improvement is needed to have a better representation of the phenomena occurring on the wing. In the second part, the unsteady panel method is applied to computing the flow around two airfoils moving relative to each other. The outcome of the computation is shown, together with a comparison between the forces obtained in the single and multiple airfoil flapping motions.

The analysis is focused on the forces produced by airfoils with different thickness undergoing pitch and plunge motions with different amplitudes. The results are compared with the linear theory of Theodorsen and Garrick [34, 36] and another potential flow model [3, 4, 5, 6, 7, 8, 9, 10, 11, 22]. The computations are performed in time domain with a time step much smaller than the minimum required. In particular 100 samples per cycle have been used for all the cases presented.

Before carrying out the comparisons, the effect of the spatial discretization has been assessed. The airfoil surface is split in panels introducing points. The panels that connect the points are assumed flat and the quantities related to each panel are constant over the panel itself. Three airfoils of the same family with different thickness have been considered: NACA 0001, 0006 and 0012. Only plunge motions have been considered at this stage. A convergence study has been performed to assess how the forces vary increasing the thickness. Figures 4.15 and 4.16 show that the thickness does not affect the lift and the pitching moment on the airfoil. Figure 4.17 shows some deviation in thrust between the 6% and 12 % thick and the 1 % thick airfoils. To assess whether this is due to the physics of the problem, the number of points on the airfoil has been increased to have a better resolution. The airfoils have been discretized using 60, 100, 200 and 300 panels. Figure 4.18 shows that employing a finer mesh the results drastically improve, with the thrust coefficient of the finest mesh being very close to the thrust coefficients of NACA 0006 and NACA0012. The trend

of the thrust coefficient shows that the deviation present in figure 4.17 would vanish if the number of points is further increased. This can be explained with the dependence of the thrust on the values of the pressure coefficient in proximity of the leading edge. The axial component of the force is greatly affected by the resolution at the leading edge of the airfoil, whereas the normal force depends mostly on the low-curvature part of the wing. Therefore a coarse mesh would not affect as much the lift as it affects the thrust. The 4 horizontal lines in figure 4.18 are drawn in correspondence of the maximum value of the thrust coefficient after a steady oscillation has established. If the line on top is assumed to represent the exact value of the thrust coefficient (300 panels), then the errors related to the others are, in percentage terms: 63% (30 panels), 41% (100 panels) and 9% (200 panels). Even though the convergence is not attained with 300 panels yet, it can be said that the exact value of the thrust is well represented with 300 panels, as the error is less than 10%.

From figures 4.15, 4.16, 4.17 and 4.18 it can also be said that in plunge the thickness of the airfoil does not matter, as the forces produced by the three airfoils are the same. Therefore in the following results the thickness is not considered anymore when the airfoil is undergoing plunge motions. Besides, it is interesting to notice that the frequency of the thrust is twice the frequency of the lift and of the flapping motion. This means that during the flapping cycle the maximum thrust is reached twice.

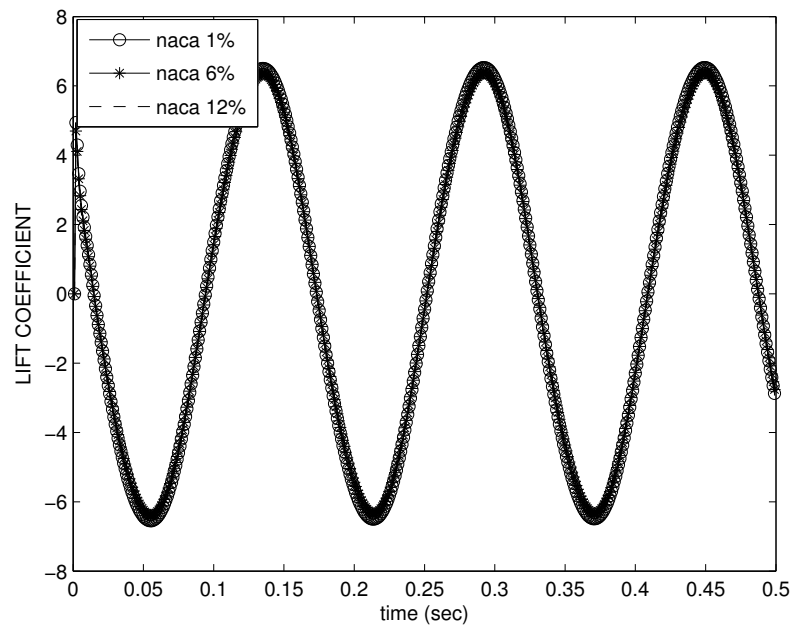


Figure 4.15: Lift vs time for 3 different thicknesses

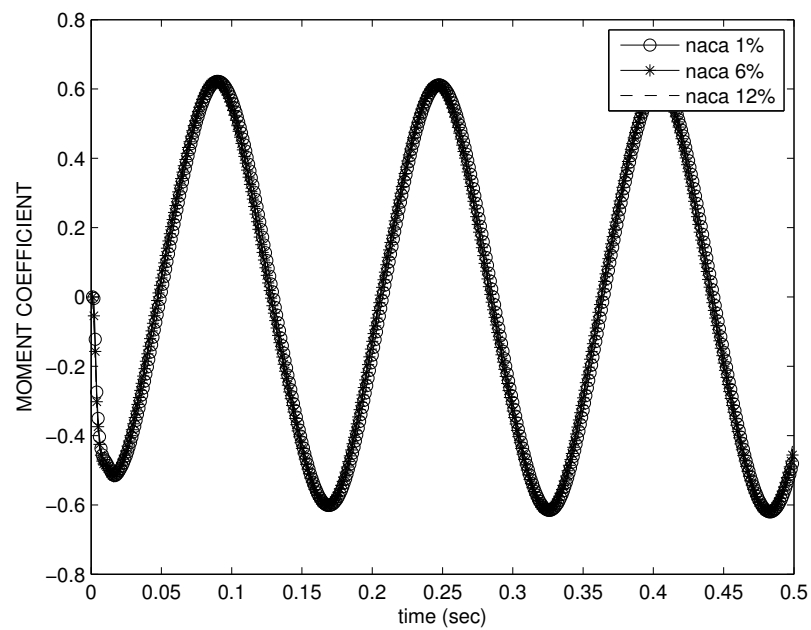


Figure 4.16: Moment vs time for 3 different thicknesses

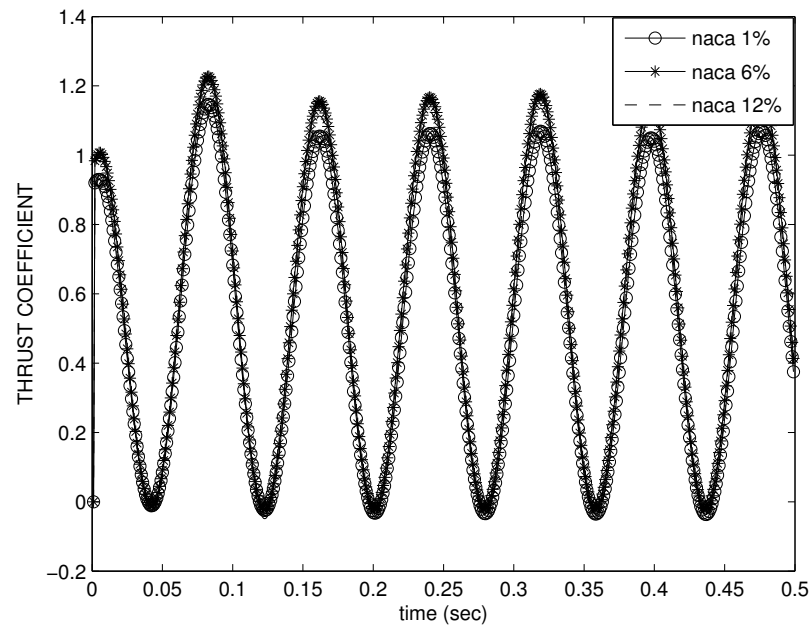


Figure 4.17: Thrust vs time for 3 different thicknesses

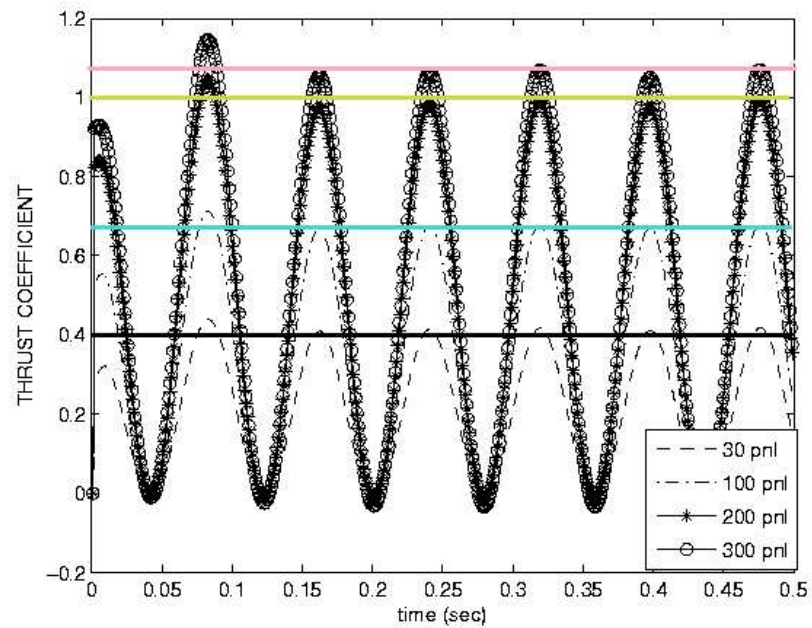


Figure 4.18: Thrust vs time for coarse, medium, fine and very fine mesh

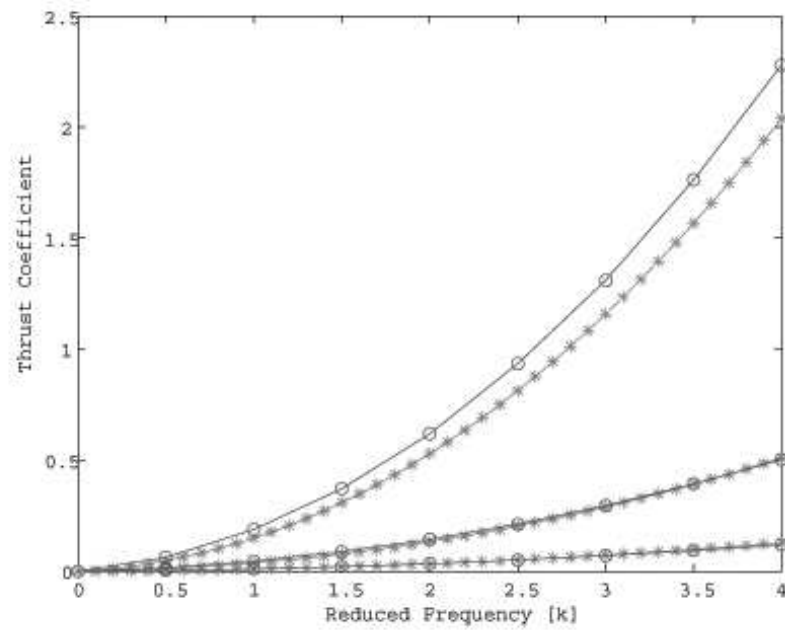


Figure 4.19: Thrust Coefficient versus reduced frequency for $h = 0.1, 0.2, 0.4$

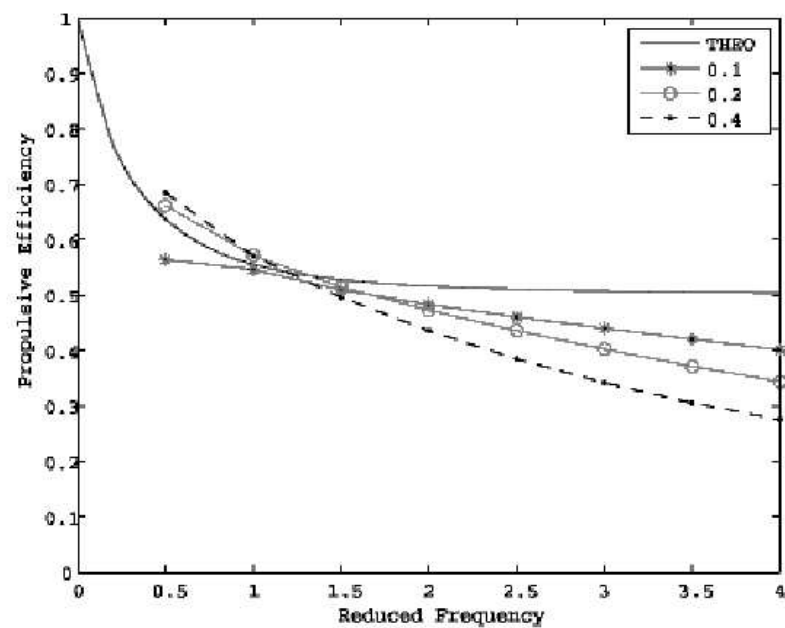


Figure 4.20: Efficiency versus Reduced Frequency for $h = 0.1, 0.2, 0.4$

The effect of the reduced frequency on the forces produced by the airfoil and on the wake generated is investigated in more detail. It would be difficult to quantify this effect showing pictures of the thrust and the lift in time domain. A parameter that is largely used for the analysis of this dependence is the value of the thrust coefficient averaged over the flapping cycle. The mean lift coefficient is zero, as the same lift that is yielded in the downstroke is produced in the upstroke as well, giving a zero net contribution over one flapping cycle. Young [83] uses the maximum value of the lift, that can be compared with Theodorsen [34]. In the present investigation the thrust coefficient is used to measure the propulsive force generated by the flapping airfoil. For plunge motions in inviscid flows Garrick [36] showed that the mean thrust coefficient is positive for all the frequencies. This can be explained theoretically with the concept of effective angle of attack. Knoller [38] and Betz [39] proved it showing that in a sinusoidal plunging motion the velocity seen by the airfoil is the sum of the forward and the freestream velocity. As the force produced must be normal to the velocity, the force vector is canted forward. This will result in a zero mean lift over the cycle, but a positive thrust (see also Jones [23]). In figures 4.17 and 4.18 the minimum value of the thrust is not zero, as it should be from the theory, but it stays below zero. This is due to numerical inaccuracy, but it does not affect the computations. The analytical expression found by Garrick [36] for the mean thrust coefficient is:

$$C_t = \pi (kh)^2 (F^2 + G^2) \quad (4.1)$$

Equation 4.1 shows that the thrust depends on the square of the reduced frequency and amplitude. Another important parameter for the evaluation of flapping wing performance is the propulsive efficiency, for which an analytical expression has been found for plunge motions:

$$\eta = \frac{F^2 + G^2}{F} \quad (4.2)$$

Figures 4.19 and 4.20 show the results of the numerical computations carried out with the UPM developed in the present investigation. Figure 4.19 represents the mean thrust coefficient versus the reduced frequency for 3 different amplitudes of the plunging motion. The amplitudes are in percentage of the chord of the airfoil, which is taken as the unitary length. The scale of the plot does not allow to spot the deviation between theoretical and numerical results for the lower amplitudes. However, the curves obtained with the highest amplitude show that the higher the frequency of oscillation, the larger the difference between Garrick and UPM. It is straightforward to attribute this difference to the out-of-plane motion of the wake in the UPM. As shown by Jones and al. [8], if the Unsteady Panel Method is run with the wake prescribed, the deviation between the two models vanishes.

Similar conclusions can be drawn observing figure 4.20, in which the propulsive efficiency of the airfoil is depicted for the same computa-

tions of the thrust coefficient. The efficiency is an important parameter in the flapping motion, as it gives a measure of the output compared to the input. The expression used in the UPM to compute this parameter is:

$$\eta = \frac{TU_\infty}{L\dot{h} + M\dot{\alpha}} = \frac{C_t}{C_l\dot{h}/U_\infty + C_m\dot{\alpha}c/U_\infty} \quad (4.3)$$

The last expression is obtained by dividing numerator and denominator by $\frac{1}{2}\rho U_\infty^3 c$ and simplifying where it is possible. The plot show that the efficiency decreases with the increase of the reduced frequency. Besides, even though the efficiency predicted by Garrick is not dependent upon the amplitude of the motion, the numerical predictions show a strong dependence upon it. Figure 4.20 shows that the higher the frequency, the smaller the efficiency. The amplitude of the motion has a negative effect: increasing the plunging amplitude, the efficiency decreases due to the wake convection. This indicates the loss of energy in the flapping motion due to the shedding and the convection of the vortices behind the airfoil. An important remark that must be made concerns the ideal flapping motion: based on the thrust coefficient, the most convenient way to flap the wing is increasing the frequency and the amplitude of the motion. If the efficiency is considered as well, a trade-off is necessary to achieve acceptable values of efficiency. The parameters analyzed and shown in this paper are based on inviscid flow assumption. Therefore the viscous effect are not taken into account. The loss in efficiency with the increase of the frequency is

due to the out-of-plane displacement of the wake. Even if the wake is bound to lie on a plane behind the wing, there is a drop in efficiency when the frequency increases. This drop follows from the vorticity shedding process. When the viscous losses are added on top of the inviscid ones, the drop in efficiency is even larger.

Next, the same kind of results is discussed for pitching airfoils. The pitch motion needs two parameters to be defined: the amplitude and the point about which it is occurring. In the following computations, the amplitudes are respectively 2, 5 and 10 degrees, whereas the points chosen for the rotation are the leading edge and the aerodynamic centre of the airfoil. The frequency range investigated is wider than in plunge for comparison with the results. The thrust coefficients and the propulsive efficiency are still taken into account to evaluate the performance of the pitching motion.

In figure 4.21, the rotation occurs about the leading edge with an amplitude of 2 degrees. NACA 0001, NACA0006 and NACA0012 are used for the computations. In particular, it is worth noticing the contribution of the thickness of the airfoil in the pitching motion. The thinner the airfoil, the greater the agreement with the linear theory. It is also worth outlining that in this case the numerical predictions lie below the theoretical ones. Unlike in plunge, in pitch the thrust coefficient is not positive for all the frequencies. It is negative up to $k = 2 - 3$, and then becomes positive with a parabolic trend as in plunge. The negative values of the thrust for low frequencies are still

obtained under inviscid flow assumption. There is no viscous drag affecting these values.

Figure 4.22 shows the efficiency of a NACA0012 airfoil undergoing a pitch motion about its quarter chord point. Three amplitudes are investigated to assess the trend of the efficiency versus the reduced frequency, with the amplitude as parameter. The graph shows that, unlike the plunge case, in pitch the efficiency has a peak at a reduced frequency of about 5. For low values of frequency it is negative, due to the negative values of the thrust coefficient. It increases very sharply up to the peak point and then steadily decreases for high frequencies. It is remarkable that also in this case the amplitude of the motion significantly affects the efficiency: the greater the amplitude, the lower the efficiency.

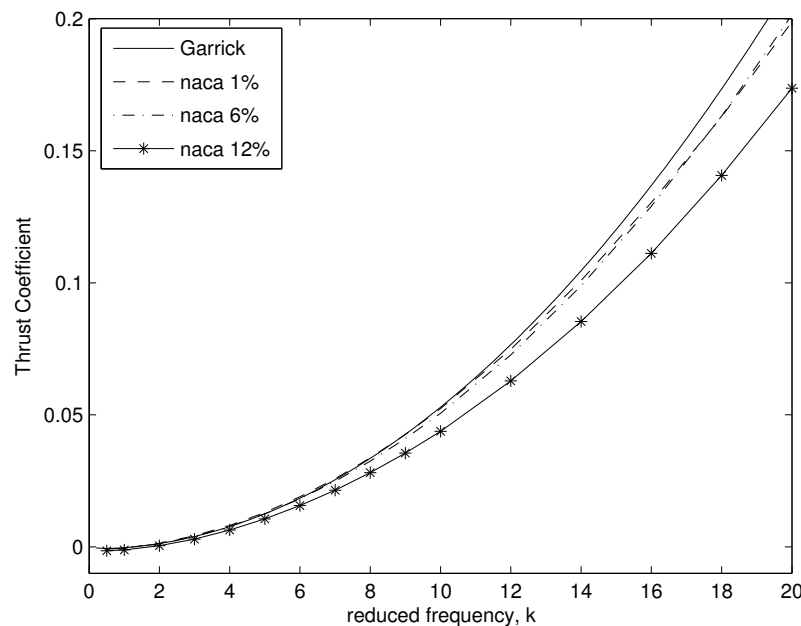


Figure 4.21: Comparison of thrust coefficient for pitch

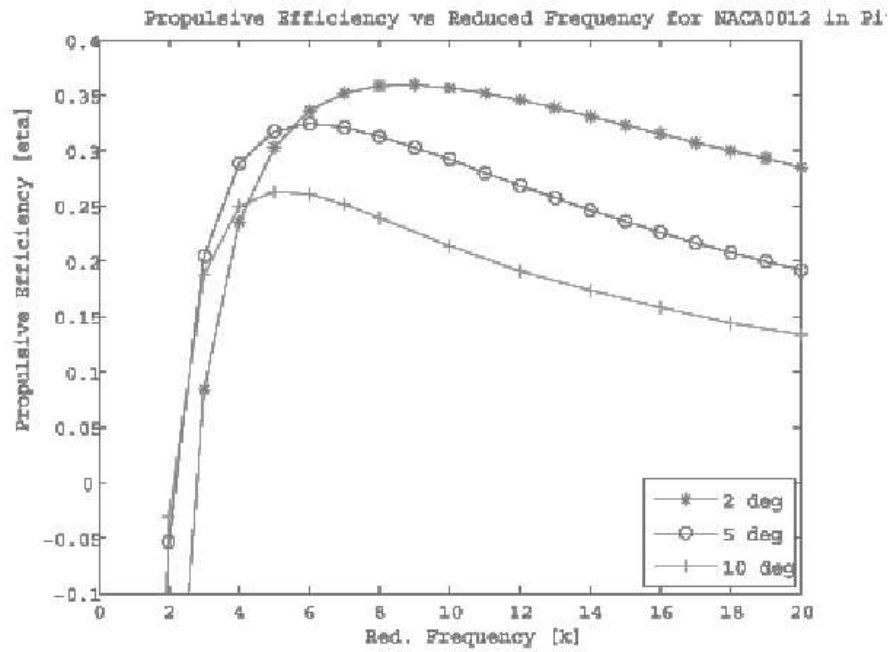


Figure 4.22: Comparison of efficiency coefficient for pitch motions

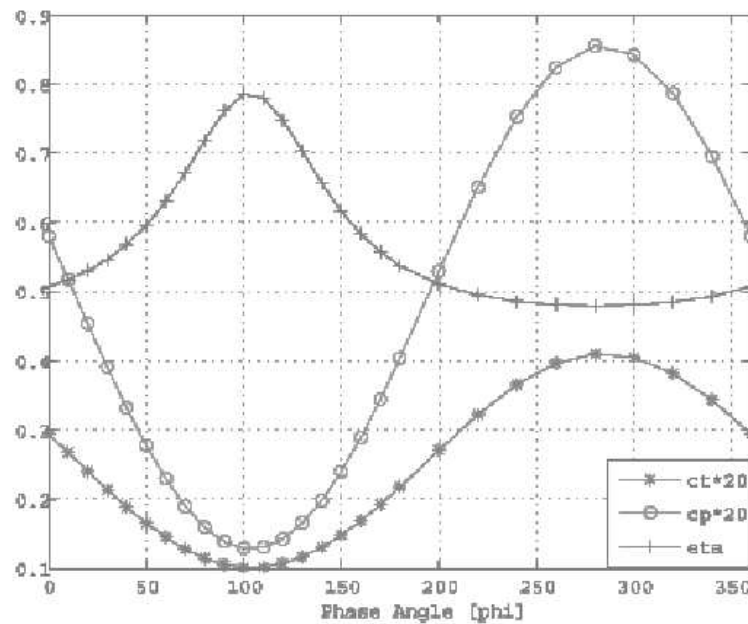


Figure 4.23: Efficiency, thrust and power coefficients for combined motions

After computing pitch and plunge motions separately, tests with combined motions have been performed. Another important parameter, the phase angle between the two motions, adds up to the analysis. In the present paper, the phase angle is the only parameter that has been taken into consideration for the following computations. The amplitudes of the plunge and pitch are respectively 0.2 and 4 degrees. The pivot about which the airfoil is rotating is the quarter chord. The airfoil investigated is a NACA0012 and the reduced frequency is 0.5. This value is significantly lower than those considered in the rest of the paper and it is chosen to compare with Jones and al. [8]. The phase angle range investigated spans the whole 360 degrees and the pitch motion is assumed to follow the plunge.

Figure 4.23 shows the results of the computations. The thrust coefficient and the power coefficient are multiplied by a factor of 20 to fit in the same plot with the efficiency. It is immediately worth noticing the peak in the efficiency. In particular the angle at which this is achieved lies between 90 and 100 degrees. The maximum value of the efficiency is almost 80 %, slightly lower than the value predicted in Jones [8]. This is higher than the value obtained plunging the airfoil at the same frequency. In particular there is a range of phase angles in which the efficiency is higher than in plunge. Beyond this range, the efficiency drops below the one obtained with the plunge motion. The thrust and the power coefficients vary sinusoidally with the phase angle, as indicated in the figure. It can be asserted that it is convenient to combine pitch and plunge as long as pitching follows plunging of about 90

degrees or so.

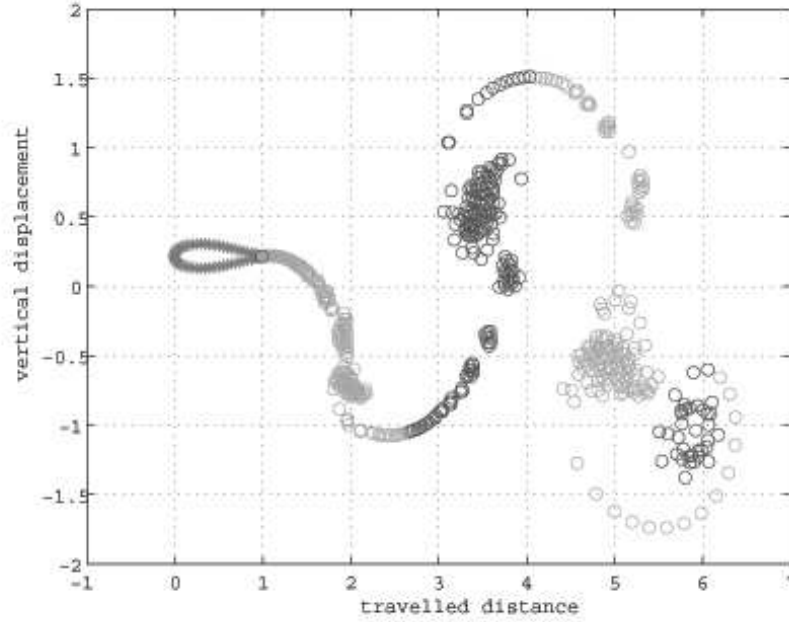


Figure 4.24: Wake Convection for $k=2$, $h=0.4$

So far the attention has been focused on the aerodynamic forces produced by the airfoil, combining these to obtain global parameters as efficiency and thrust coefficient. It has been said several times that the difference between the model proposed in this paper and the linear theory is mainly the convection of the wake, but no wake visualization has been given. Figures 4.24 and 4.25 show how the assumption of flat wake is wrong when the amplitude and the frequency of the motion are large. In particular, the amplitude in the two pictures is the same and equal to 0.4 times the chord, whereas the frequency is changed from 2 in fig. 4.24 to 3 in fig. 4.25. Increasing the frequency of oscillation, the roll-up occurs earlier and the out-of-plane displacement is greater, affecting the forces on the airfoil. Fairgrieve and DeLaurier [40] developed a model in which the wake could be convected as a flat

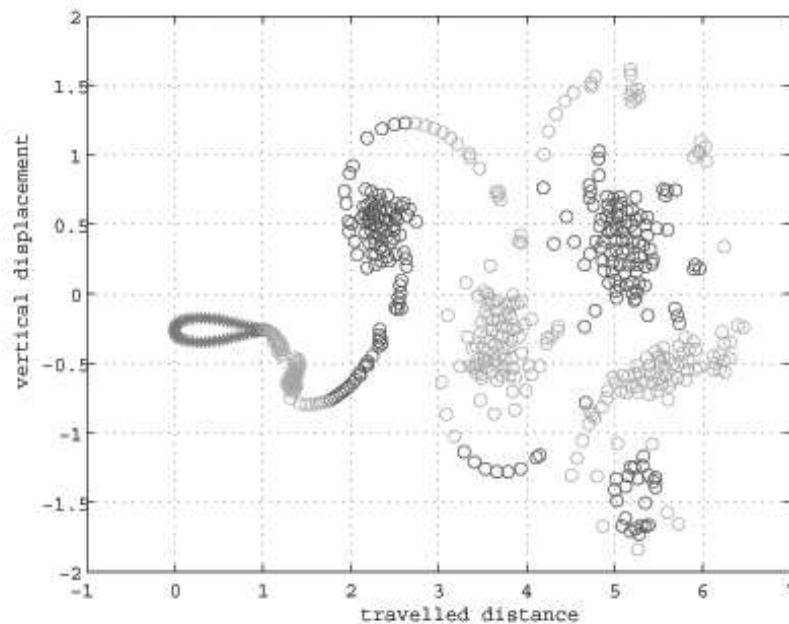


Figure 4.25: Wake Convection for $k=3$, $h=0.4$

surface, fixed in space where it is shed and free to move. Jones and Platzer [8] also show that the Unsteady Panel Method yields results much closer to the theoretical predictions when the wake is bound to a plane behind the airfoil. Therefore the effect of an accurate wake convection is not neglectable when the assumptions of small oscillations are violated.

4.3 Multiple Airfoil Flapping

Jones and Platzer [7] showed the advantages of having a double airfoil configuration to increase the efficiency of the flapping motion. In particular they focused their attention on the Ground Effect, which they exploited in the prototype built at the Naval Postgraduate School

(NPS). Based on their extensive work in the assessment of the performances of double airfoil configurations, it is made possible to test the reliability of the code presented in this paper for the same configurations.

In this section two parameters are taken into account for comparison: thrust coefficient and efficiency. Only plunge motions are considered, because it is proved to be better than pitch in the frequency range investigated. For the opposed flapping combination (GE), the mean distance separating the airfoils' chords is 1.4 times the chord and they are plunging with an amplitude of 0.4 times the chord. For the Schmidt Propeller (SCHMIDT), the horizontal distance between the two leading edges is 1.2 times the chord. This means that the trailing airfoil's leading edge is 0.2 times the chord behind the leading airfoil's trailing edge. In this case the amplitude of the motion of the leading airfoil is 0.4 times the chord, whereas the trailing airfoil is travelling steady downstream. For comparison with the results of Jones and Platzer [7], the frequency ranges from 0 to 1.

Figure 4.26 shows the thrust coefficient of the two configurations compared with the linear theory of Garrick. Figure 4.27 shows the comparison of the efficiency. Regarding the thrust, the comparison is made averaging the thrust coefficients coming from the two airfoils. In SCHMIDT, the c_t is lower than Garrick, which is lower than single airfoil flapping, as shown in fig. 4.19. This is due to the very small amount of thrust generated from the trailing airfoil. In GE, the force

produced by either of the airfoils is the same for symmetry, therefore there is no need for averaging. The thrust yielded with GE is the highest achieved with all the configurations, showing that the forces increase reducing the distance between the two airfoils.

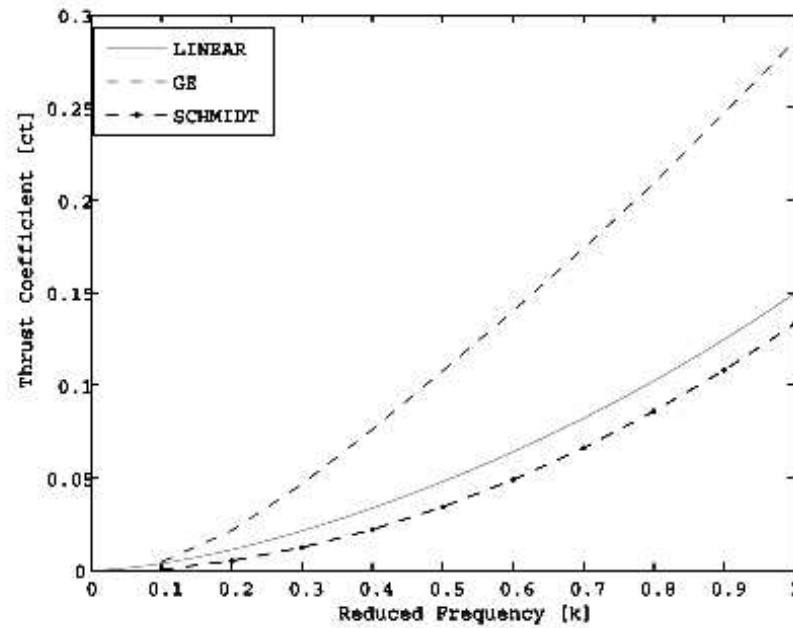


Figure 4.26: Thrust Coefficient of GE and SCHMIDT vs Garrick

Regarding the efficiency, fig. 4.27 shows that both combinations yield higher values than the theoretical predictions. In particular for SCHMIDT the efficiency is lower for very low frequencies, up to 0.3, and then becomes greater than GE. The interesting aspect SCHMIDT is that the drop in efficiency is not so significant between 0.5 and 1: this might be due to the gain yielded by the trailing airfoil. Remembering the expression of the efficiency (eq. 4.3), the input power for the motion of the trailing airfoil is zero, therefore the efficiency in this case is equal to the ratio of the total thrust and the input power of the leading airfoil, where the total thrust is the sum of the thrusts produced by

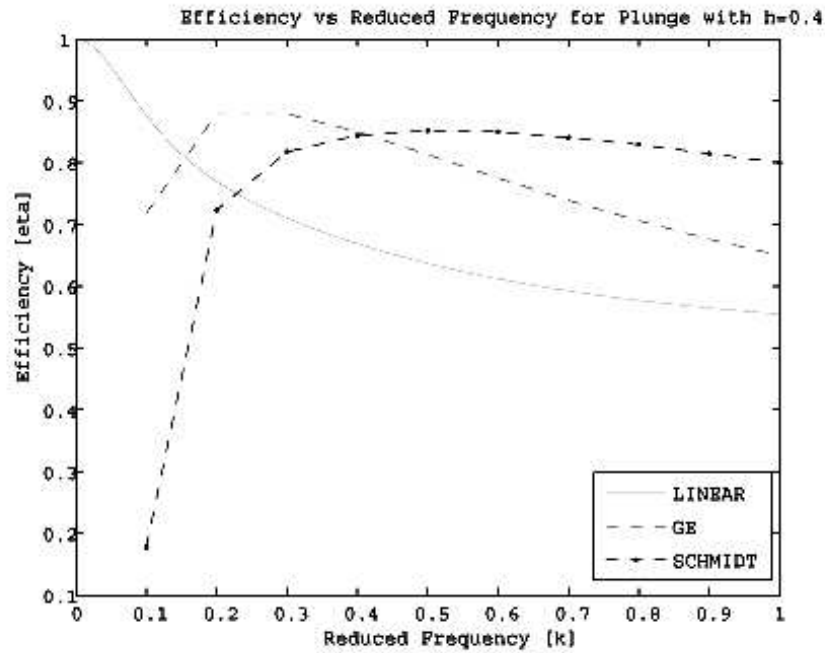


Figure 4.27: Efficiency of GE and SCHMIDT vs Garrick

the two airfoils. The GE combination yields greater efficiency than Garrick, but the drop in the frequency range investigated is greater than SCHMIDT. Considering that in this latter case the trailing airfoil operates at very low values of thrust and that the viscous losses need to be added on top of the inviscid thrust, SCHMIDT might be not as advantageous as GE, that shows lower efficiency but considerably higher thrust.

Regarding the biplane configuration, the effect of reducing the distance between the two airfoils is investigated. The motion is prescribed as sinusoidal with a reduced frequency of 3 and amplitude of 0.4 times the chord. Figures 4.28 and 4.29 show the increase in lift and thrust coefficients reducing the vertical gap from 4 to 1.4 times the chord. The extreme case of infinitely distant airfoil is equivalent to single flap-

ping, which is examined in the paper and fig. 4.27-4.26 have shown less efficient than the biplane arrangement. Therefore a real advantage can be gained reducing the distance between the airfoils due to the influence of the wake released by either airfoil on the other one. This can be equivalently expressed saying that when the wing is flapping close to the ground, the lesser the distance the larger the forces produced. Figures 4.30 and 4.31 show the difference in the convection of the wake for 2 cases with different mean distance between the airfoils. When the gap is greater, the wake is closer to the single airfoil flapping, and this also shows up in the forces produced. Reducing the distance, the influence of the wakes on each other is greater, as the symmetry condition must be fulfilled and the vortices are bound not to cross the symmetry line representing the ground.

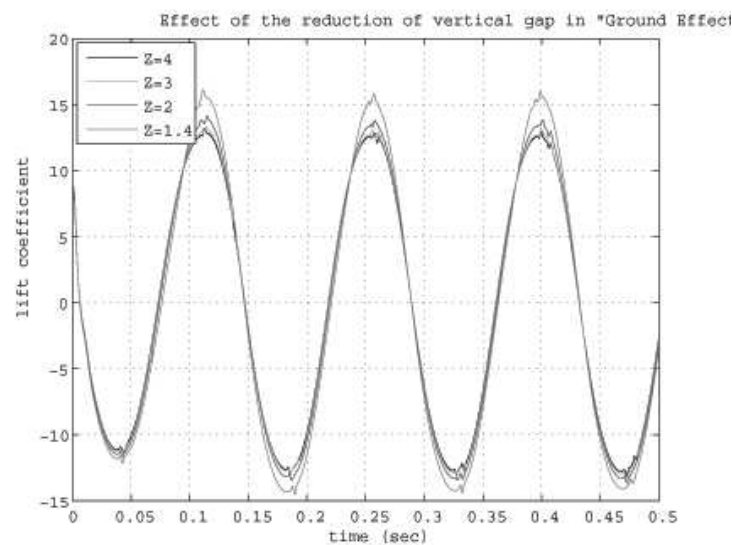


Figure 4.28: Comparison of lift coefficient for airfoils plunging at different distances

If only the wake convection is considered, the biplane combination is not challenging to simulate because of the symmetry of the problem.

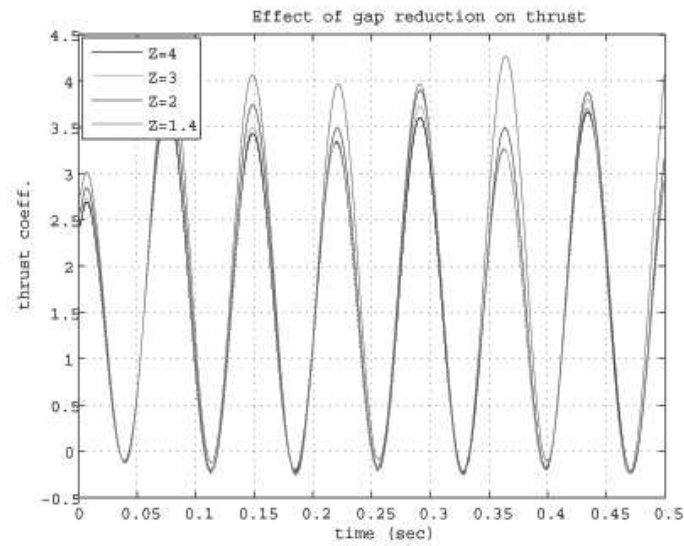


Figure 4.29: Comparison of thrust coefficient for airfoils plunging at different distances

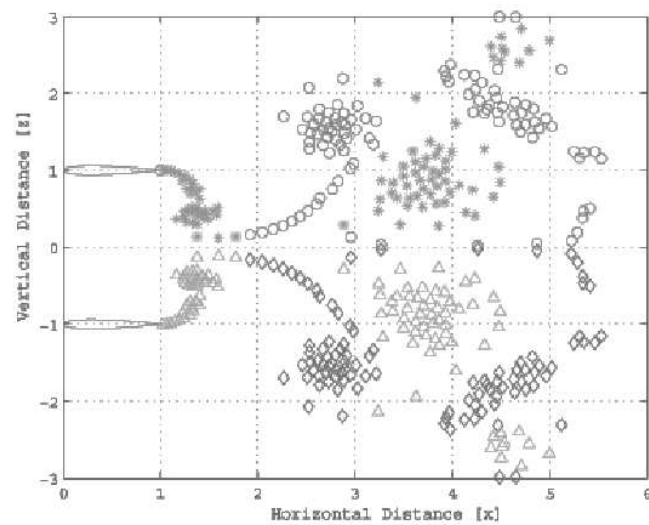
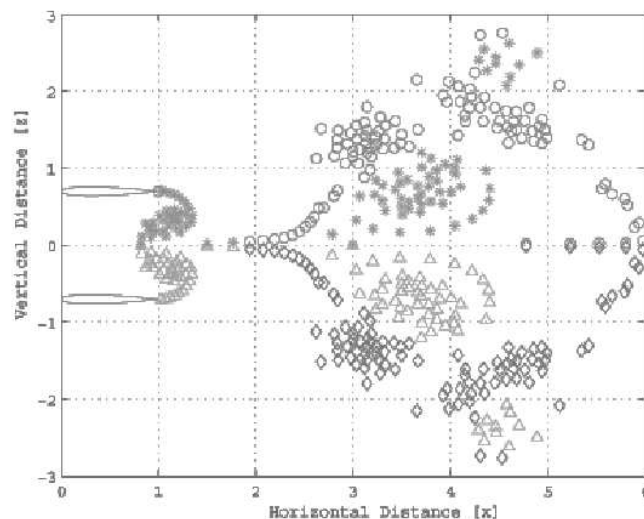


Figure 4.30: $Z=2$

Figure 4.31: $Z=3$

Even though either wake affects the convection of the other, the vortices of the two wakes never really come as close as in SCHMIDT. In the latter case the critical phase occurs when the vortices released from the leading airfoil come into contact with the surface of the trailing airfoil. Due to the approximation introduced in the algorithm for the wake convection, performed with an Euler integration scheme, the time step is bound to very small values for the vortices not to cross the airfoil's surface. Eldredge and al. [59] propose an algorithm in which, at the beginning of the generic time step, a check is performed on vortices that may have get into the airfoil. If this has occurred, the vortices are deleted from the flowfield but the circulation is still taken into account and added to the circulation bound to the airfoil surface. This algorithm has had applications in the Vortex Particle Methods (VPM) but it is not considered here for issues related to the increase in computational time. Jones and Platzer [4] propose the

use of a more accurate scheme, like a fourth-order Runge Kutta, or a reduction in time step size. They also outline the difficulty of computing the forces when the vortices are very close to the generic panel of the airfoil. In the present paper, an Euler integration scheme has been used with a very small time step (e.g. 10^{-3} sec). Besides, the aerodynamic forces acting on the trailing airfoil have been averaged in order to delete the spurious oscillations due to numerical inaccuracies in the wake convection process.

The issue with an excessive reduction of the time step is that more vortices are released in the flowfield, therefore the singularities on the trailing airfoil will be greatly affected by the closeness of these vortices. On one hand, the time step cannot be increased for accuracy problems. On the other, it cannot be reduced excessively for a smooth representation of the aerodynamic forces. Even though a few vortices may enter the airfoil over one cycle of oscillation, the thrust and efficiency in figures 4.26- 4.27 show a very good agreement with the linear theory and the numerical results used for comparison.

4.4 Vortex Particle Method

In this section the results found with the Vortex Particle Method are presented. For the sake of completeness, it is deemed useful to present the results in this context, even though the code is not complete. In particular the results that will be shown for the validation

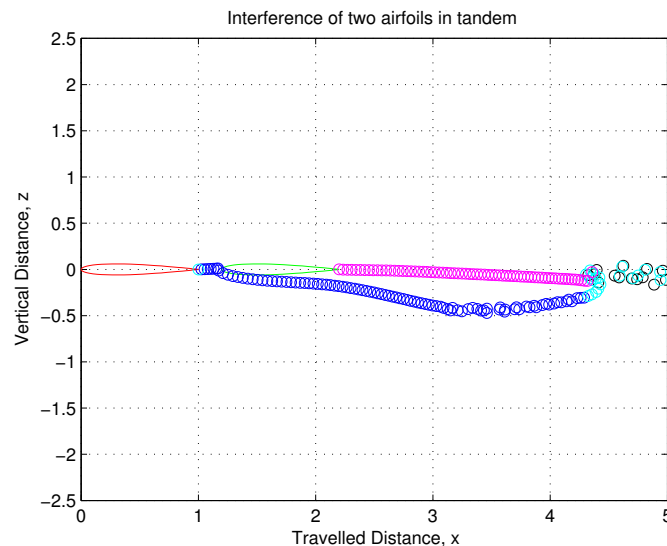


Figure 4.32: Interaction of the wake of a plunging airfoil with a trailing airfoil

of the method are only qualitative. This is due to the fact that the redistribution and diffusion schemes have not been completed and integrated in the program. Therefore the method is an enhancement of the panel method presented in the previous section, with vortices shed from each panel of the body. The viscosity is not present. Furthermore, the absence of the redistribution scheme to smooth out the vorticity contour for those particles drifting very close to the body makes it inaccurate after some time of the simulation has elapsed. Some validation results concerning the wake convection will be shown in this section, with side by side comparison with results found in literature where this is possible.

As remarked in the presentation of the VPM in the previous chapter, the core of the method is a panel method, where the wall boundary condition is enforced in order to obtain the strength of the singularities over the body. Another essential part of this method is the stage

when the strength of the particles has to be shed in the flowfield. This process requires an accurate analysis and development because the particles shed will be drifting very close to the body surface, therefore causing large errors in the value of the singularities on the surface in the next time steps. In this thesis the diffusion process has not been considered either: this is quite important to smooth out and regularize the flowfield as time elapses. Therefore from a quantitative point of view the results that are shown are not comparable to others present in literature. However, some interesting conclusions can be drawn looking at the evolution of the vortex elements around the body. It has already been remarked how the VPM can be developed from the PM. The only modification that has been done in the PM to adapt it to the VPM is to switch from doublets to vortex elements to represent the body contour. The advantage of this change lies in the form of the Kutta Joukowski condition, that lends itself well to implementation. Since the Kutta condition is formulated in terms of velocity on the panel and the relation between the slip velocity and the density of vorticity is

$$\gamma = v_\tau$$

Therefore the VPM developed in this thesis does have a panel method in order to compute the slip velocity on the panels, which is not the same panel method used in the analysis of single and multiple airfoil flapping. Before switching to the new method, the vortex method has

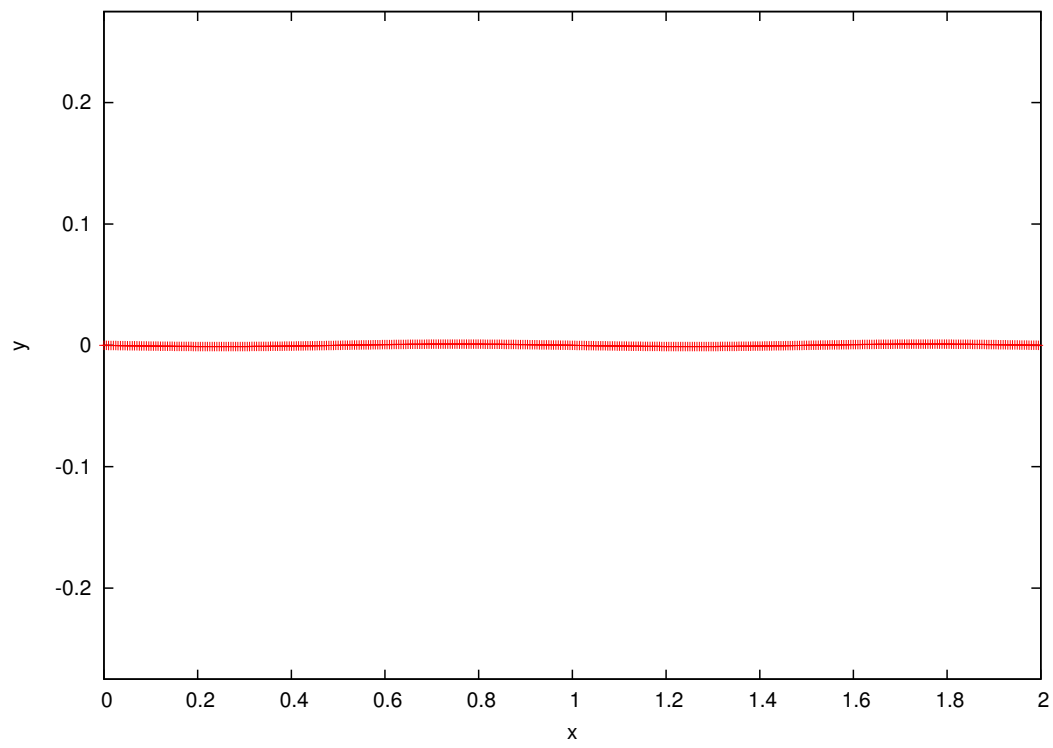
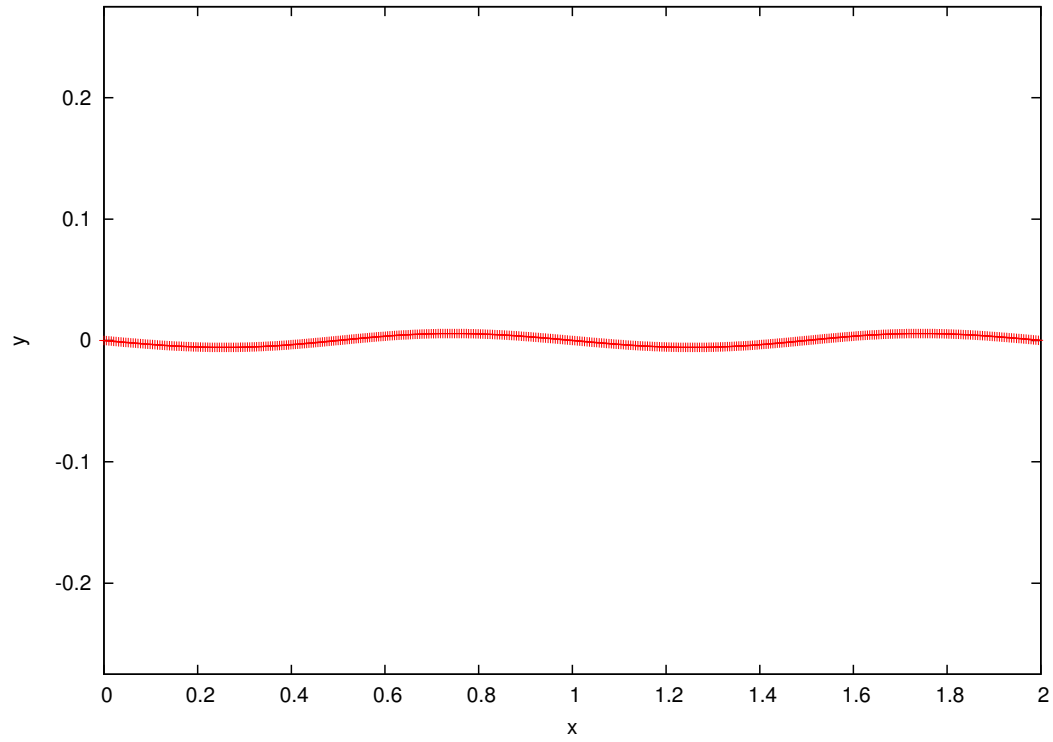
been validated against the doublet method for simple test cases. In particular the example of a 2D cylinder in a uniform incompressible flow has been considered. The comparison has been carried out considering the induced velocities on the panels. In particular, for the vortex methods they are equal to the vorticity on the panel, as shown above. For the doublet method, they can be computed using equation 3.42. The results of the comparison are not given here because they are not relevant to the rest of the thesis. The important thing is that the two panel methods are perfectly equivalent.

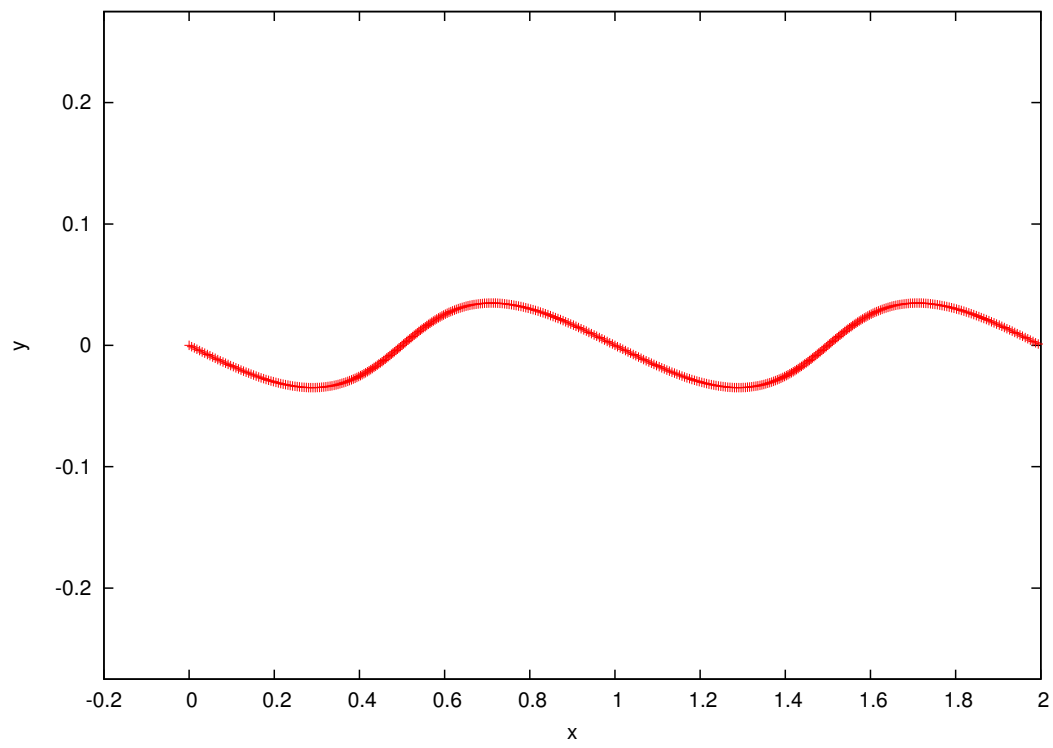
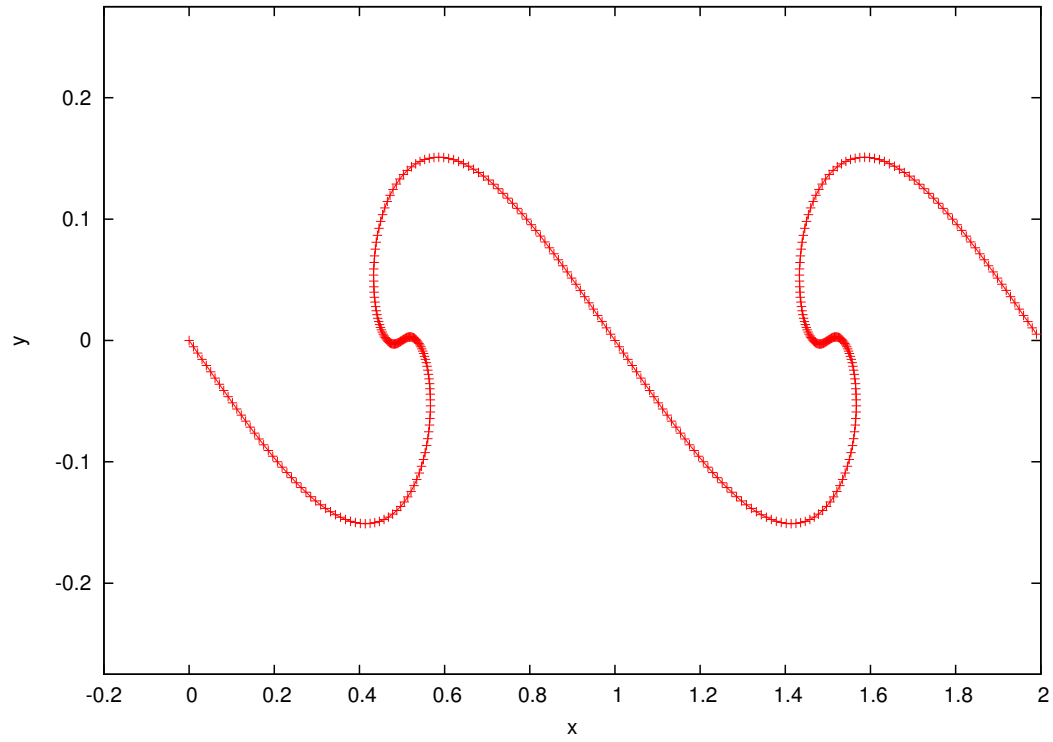
After the validation of the panel method, a few examples for the particle mollification have been followed. The choice of the kernels used to spread the vorticity is fundamental for the subsequent development of the method. The first example used for the validation is taken from Krasny ([60]). The desingularization scheme presented in his paper yields a numerically more tractable set of equations.

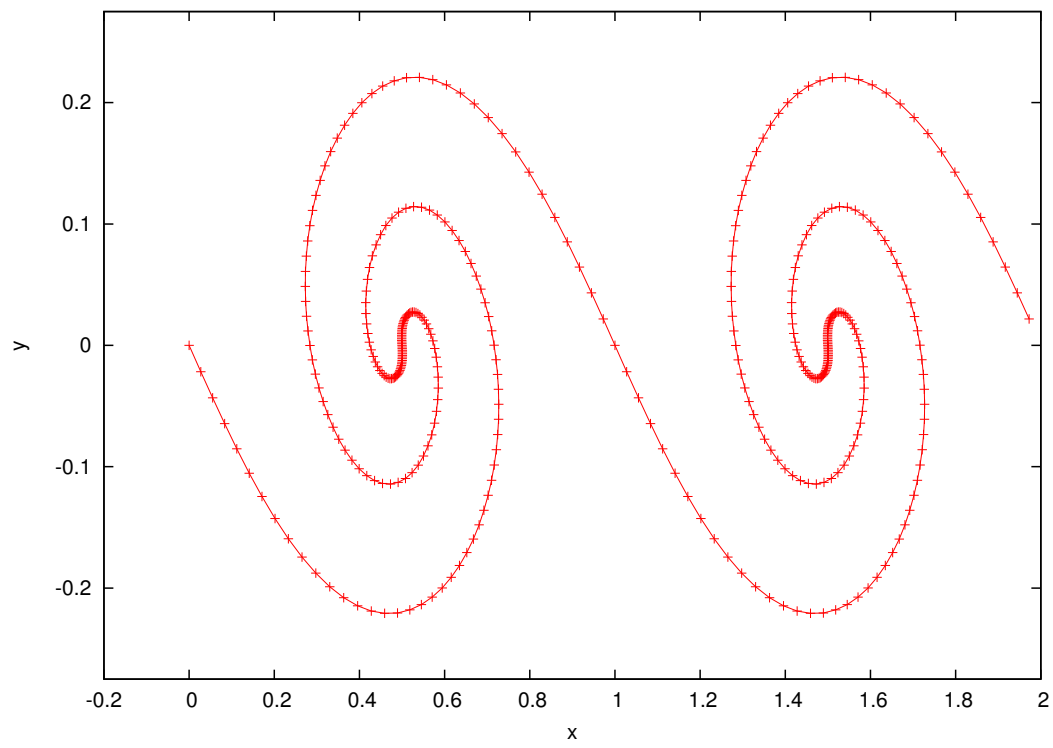
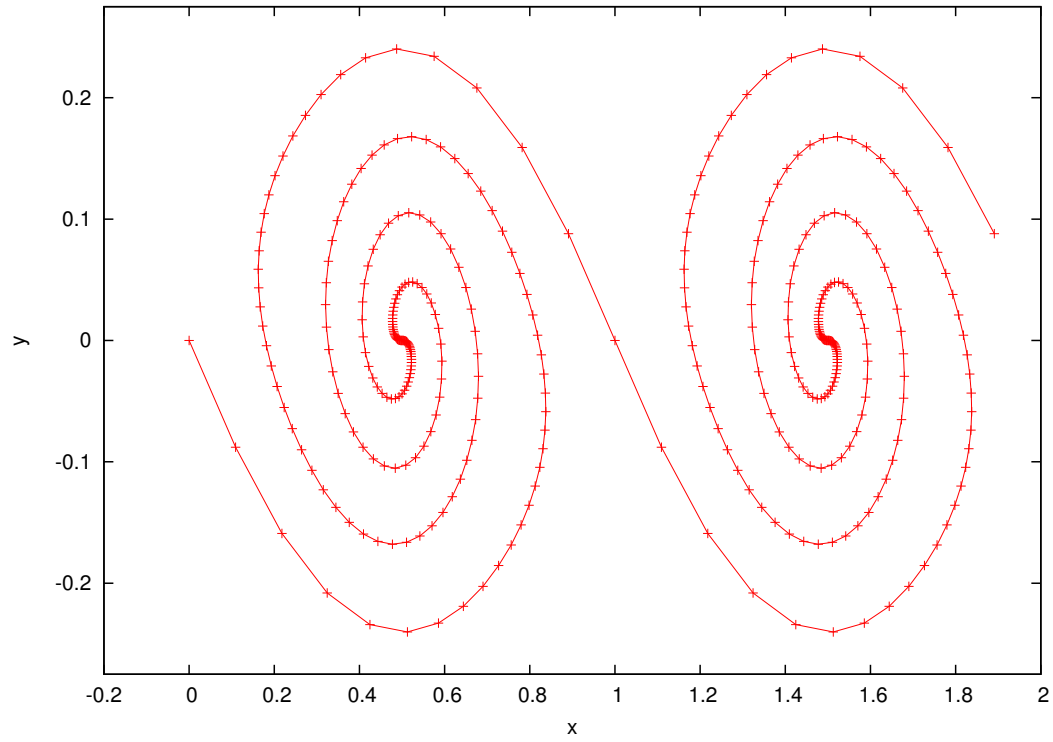
Figures 4.33-4.38 show the outcome of the computations using Krasny desingularization method. The method consists in solving with a 4th order Runge Kutta scheme the set of ordinary differential equations:

$$\begin{aligned}\frac{dx_j}{dt} &= \frac{-1}{2N} \sum_{k=1, k \neq j}^N \frac{\sinh 2\pi(y_j - y_k)}{\cosh 2\pi(y_j - y_k) - \cos 2\pi(x_j - x_k) + \delta^2} \\ \frac{dy_j}{dt} &= \frac{1}{2N} \sum_{k=1, k \neq j}^N \frac{\sin 2\pi(y_j - y_k)}{\cosh 2\pi(y_j - y_k) - \cos 2\pi(x_j - x_k) + \delta^2}\end{aligned}\quad (4.4)$$

The number of vortices used in the present case is 400. The strength of the vortices is constant and equal to 1. The initial condition describing

Figure 4.33: $t=0$ Figure 4.34: $t=1$

Figure 4.35: $t=2$ Figure 4.36: $t=3$

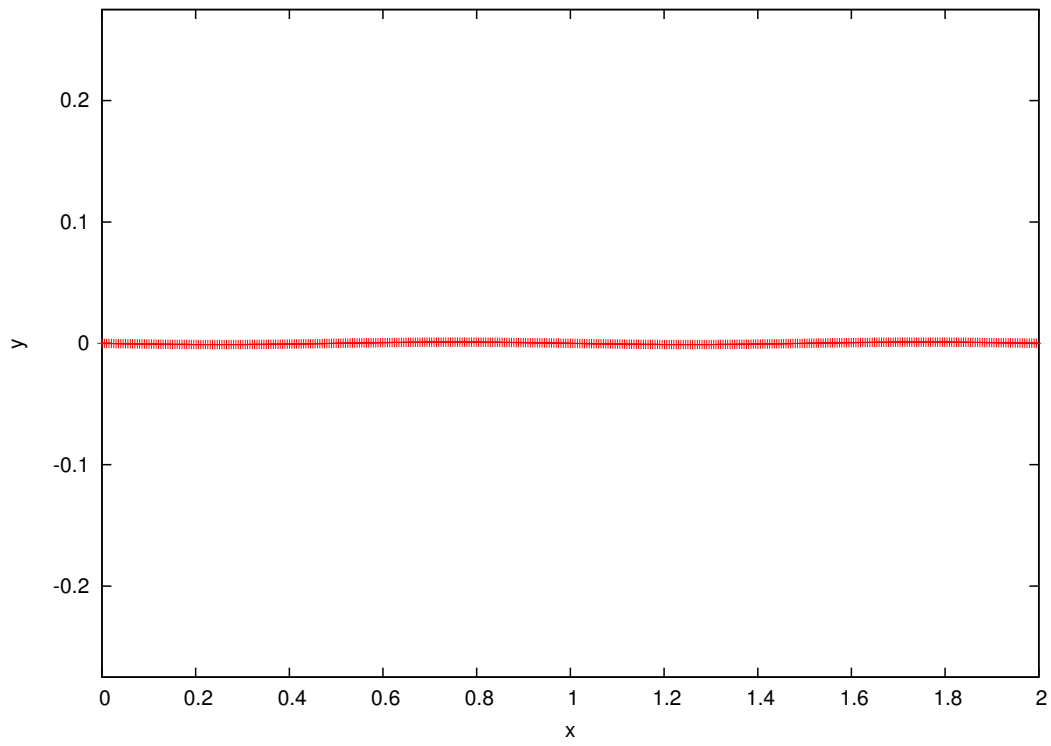
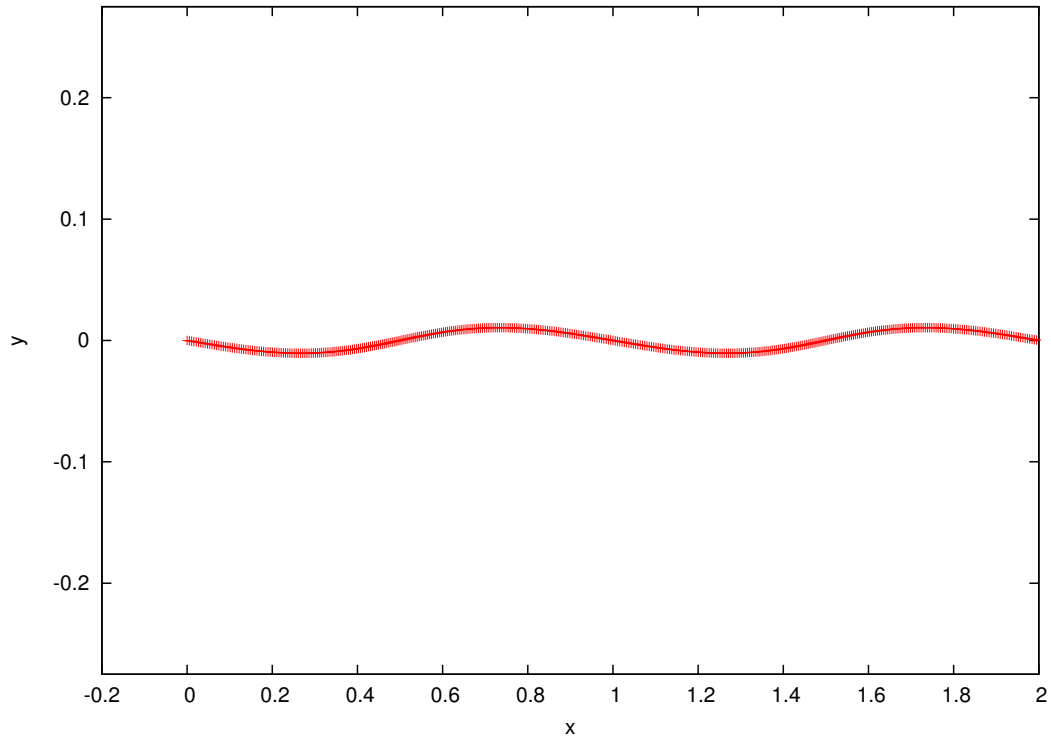
Figure 4.37: $t=4$ Figure 4.38: $t=5$

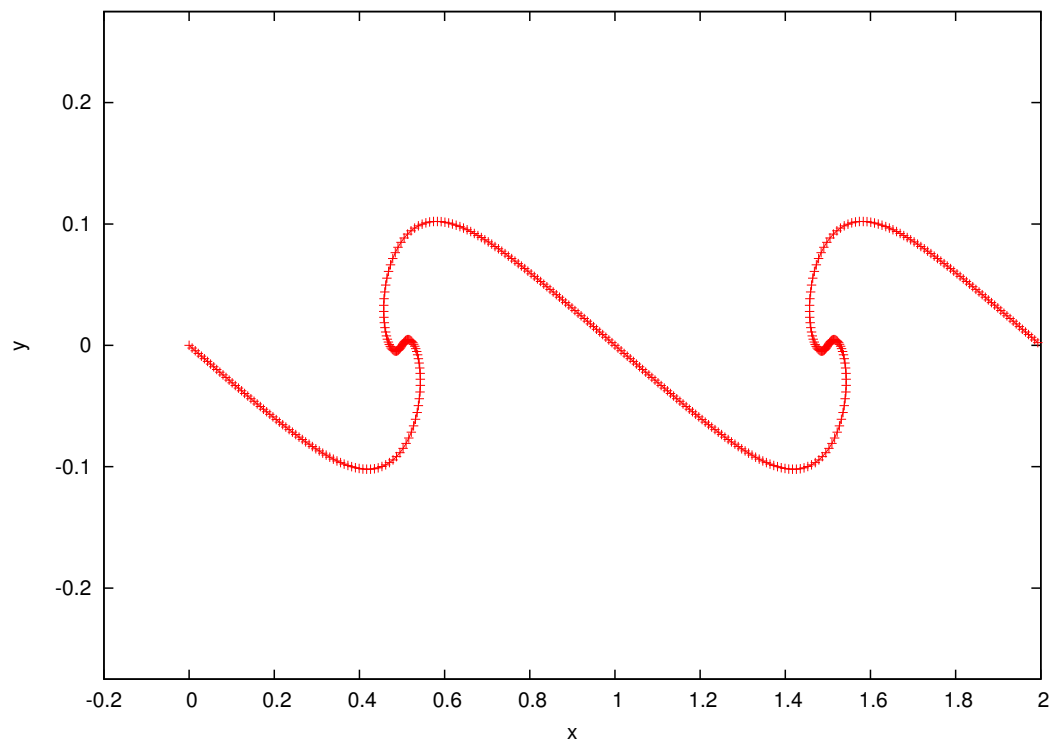
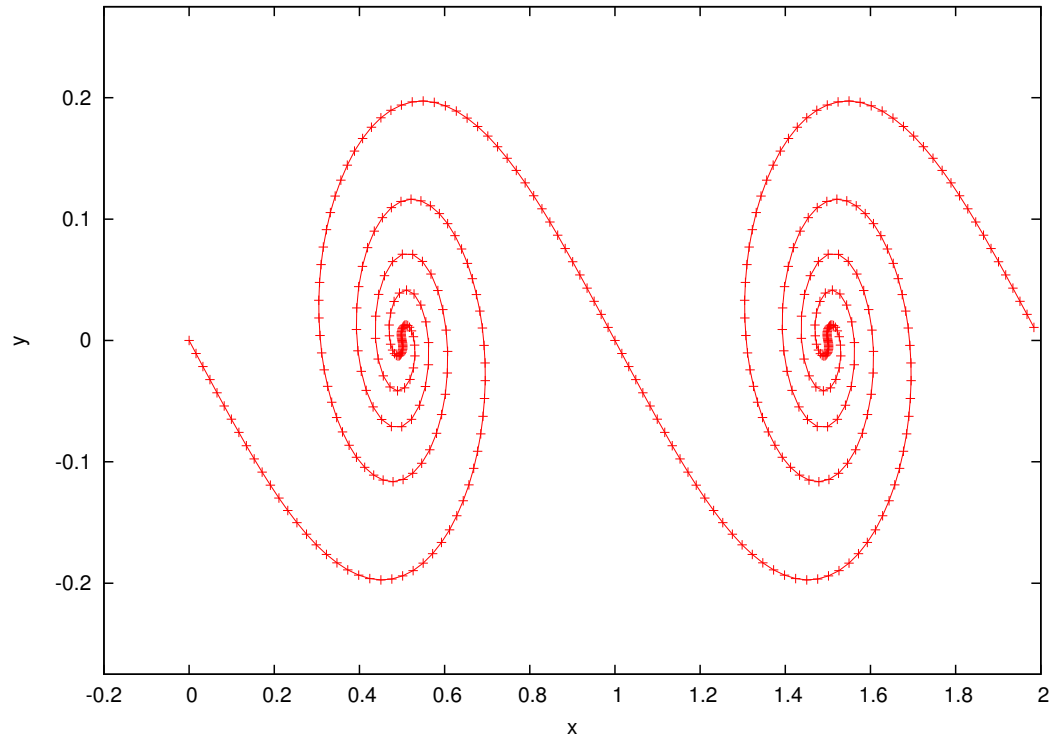
the vortex lattice shape is:

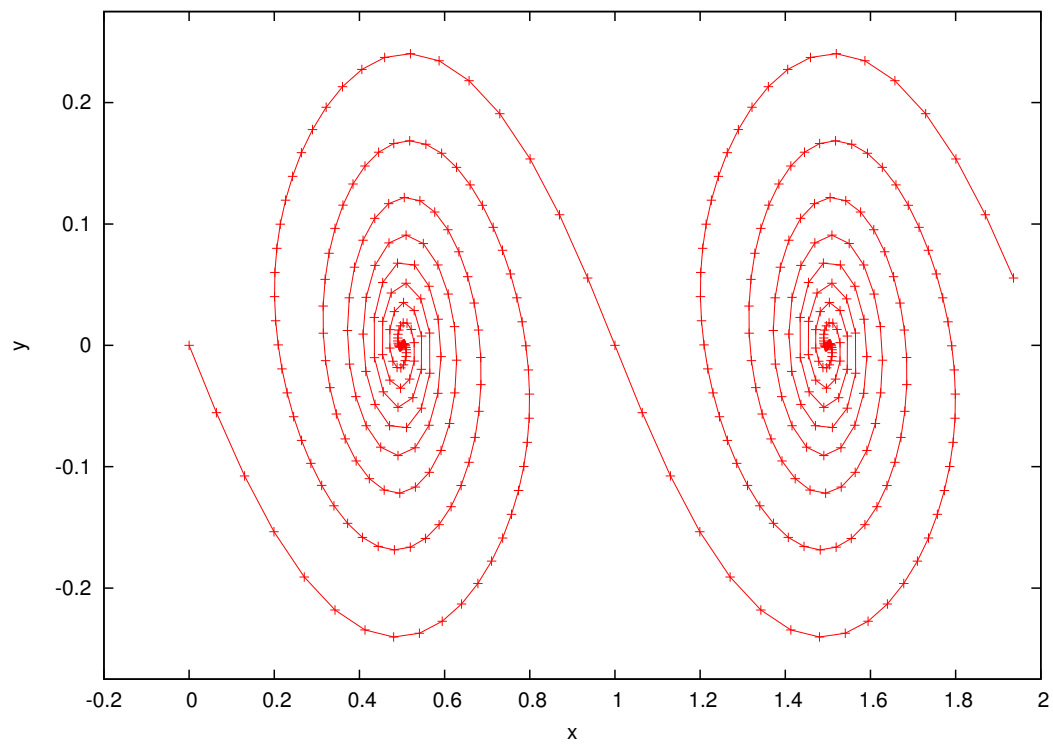
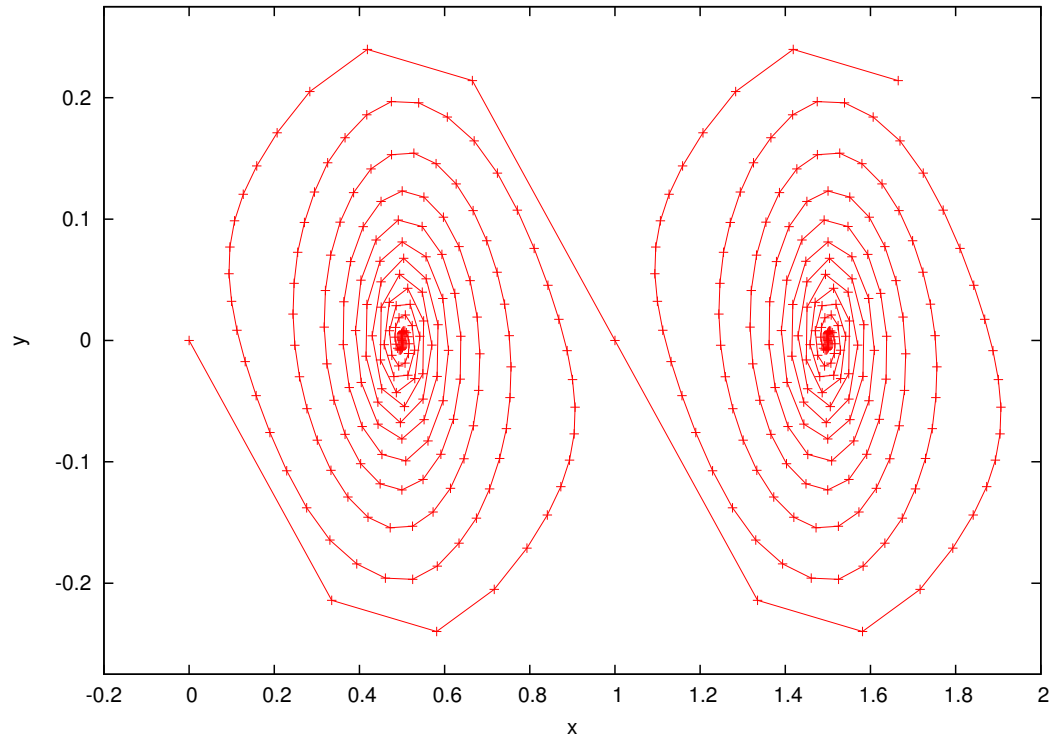
$$x_j(0) = jdx + 0.001 \sin(2\pi jdx), \quad y_j(0) = -0.001 \sin(2\pi jdx)$$

with $dx = \frac{L}{N} = \frac{2}{N}$. This gives a sinusoidal perturbation to the otherwise flat shape of the vortex lattice. The parameter that appears at the denominator, δ , has the function of desingularizing the kernel. It is strictly related to the size of the time step. The smaller δ , the smaller Δt . In the limit of $\delta \rightarrow 0$ the vortex blobs tend to become point vortices and the solution of the system becomes singular. In the present case, the use of the blob parameter allows to compute a smooth numerical solution for a longer time. The use of a smaller δ leads to a smaller time step for the integration in order to maintain the same accuracy. Figures 4.39-4.44 show the results of the computation using $\delta = 0.25$. Halving the value of the blob parameter leads to a higher value of the induced velocity in equations 4.4. Therefore the value of the time step needs to be reduced in order to obtain particle displacements of the same order as before. The vortex lattice in the second case rolls up faster, showing a greater number of turns than the first case ($\delta = 0.5$) at the same time step. The last step of the second case shows an ellipse-like shape of the core, with the ellipse major axis tilted with respect to the x -axis.

The analysis of Krasny goes on with a discussion on the stability of the solution found with the method described above. At the present time, our interest is not focused on the investigation of the solution of

Figure 4.39: $t=0$ Figure 4.40: $t=1$

Figure 4.41: $t=2$ Figure 4.42: $t=3$

Figure 4.43: $t=4$ Figure 4.44: $t=5$

the vortex lattice example. It has been shown that, with the choice of an appropriate kernel and with the introduction of a parameter for its desingularization, the solution of the convection of vortex elements can be obtained with a numerical procedure. The numerical solution tends to the analytic one for $\delta \rightarrow 0$, as demonstrated in [60].

The second example investigated concerns the convection of vortex elements initially arranged on a regular grid. The vortex blobs are placed at the centres of rectangular cells, with initial strengths given by the function:

$$\omega_0(r) = (1 - r^2)^3, \quad r = \sqrt{x^2 + y^2}$$

This function has radial symmetry, with the maximum equal to 1 in correspondence of $r = 0$, whose plot on a cartesian grid is given in figure 4.45.

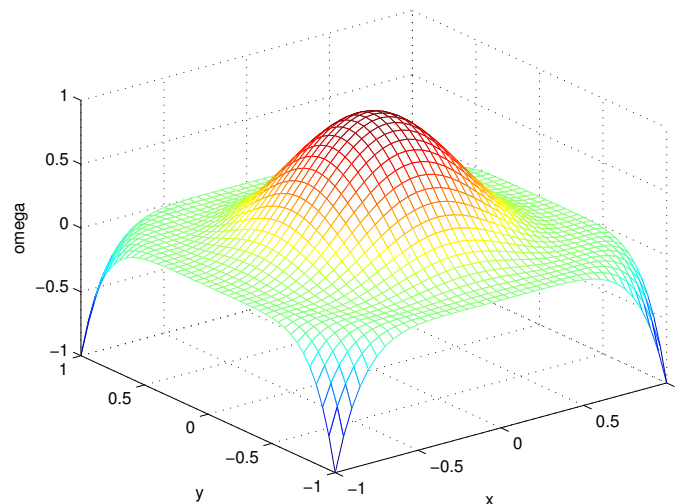


Figure 4.45: Initial vorticity distribution on a rectangular grid

The same convection scheme used in the example above has been employed in the present case. The domain used for the computation is the square with side 2 extending from -1 to $+1$ in both x and y directions. The example in here is taken from [50], with the difference that in that case the domain is elliptical, whereas we are using a rectangular grid. The particles are placed in a deterministic manner over the rectangular grid. The number of the particles used is 400, 20 on each side of the square. The mesh size is $h = 0.1$ and the core of the particles is twice the mesh size: $\epsilon = 2h$. The numerical integration of the particles trajectories is carried out using an exponential kernel instead of the kernel used in the example of Krasny.

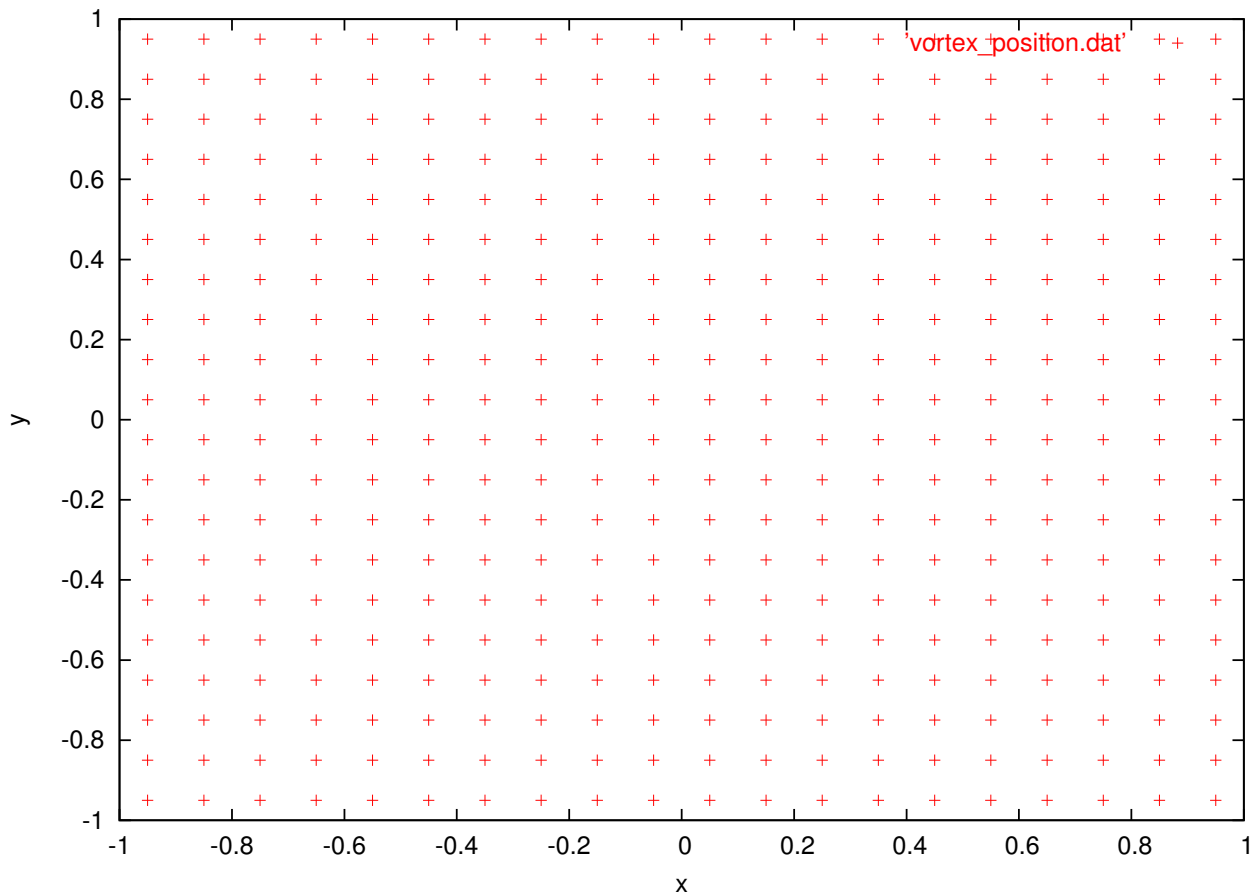
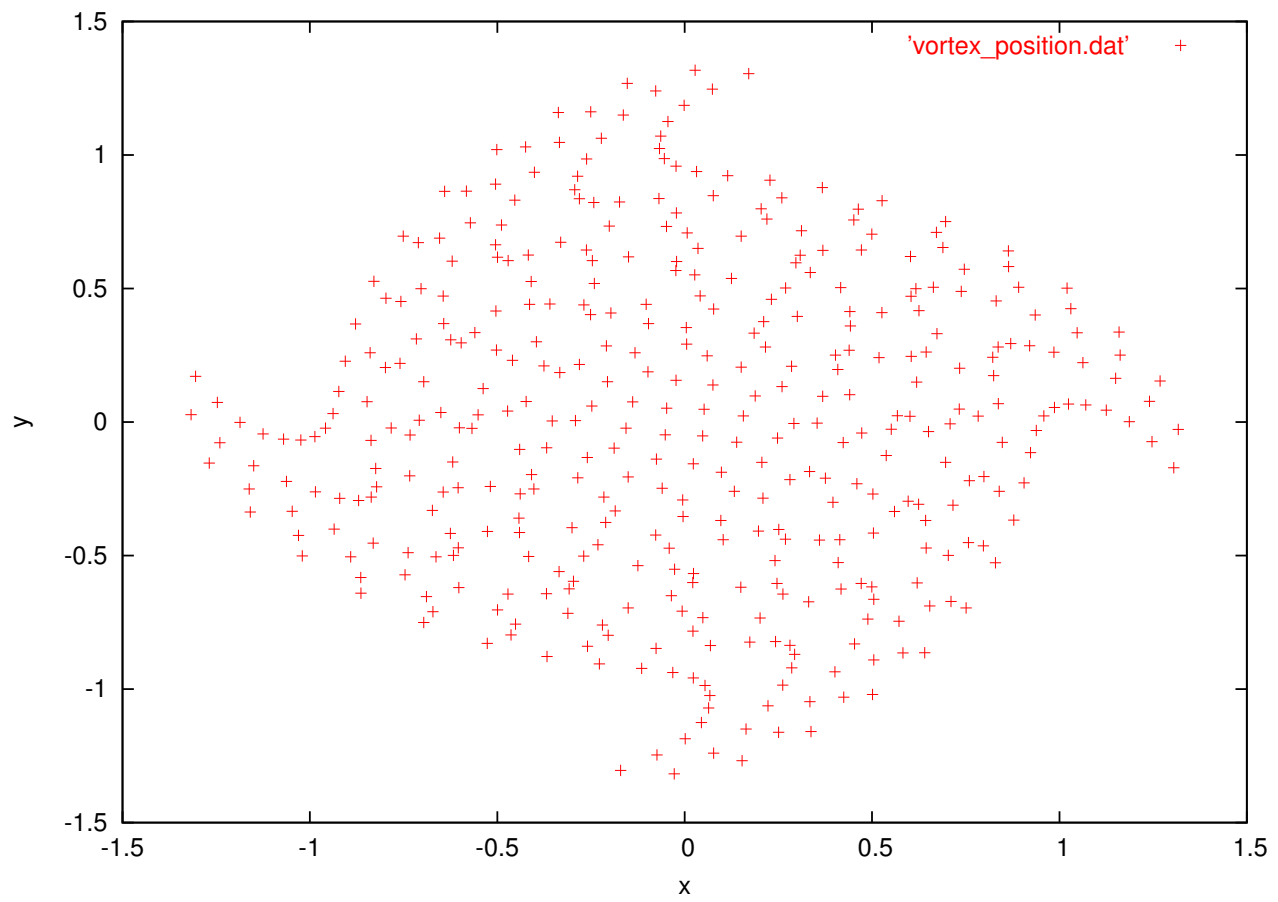
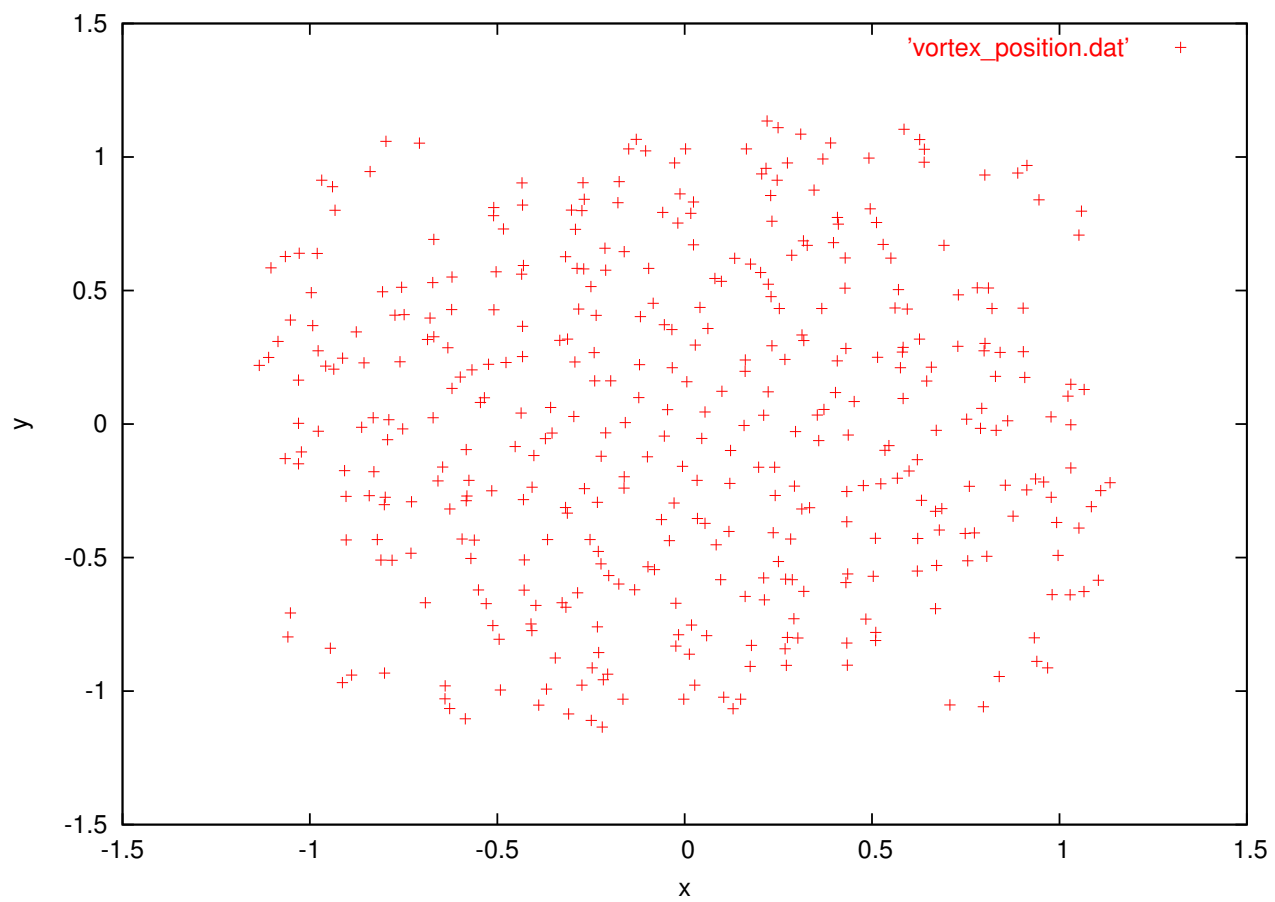
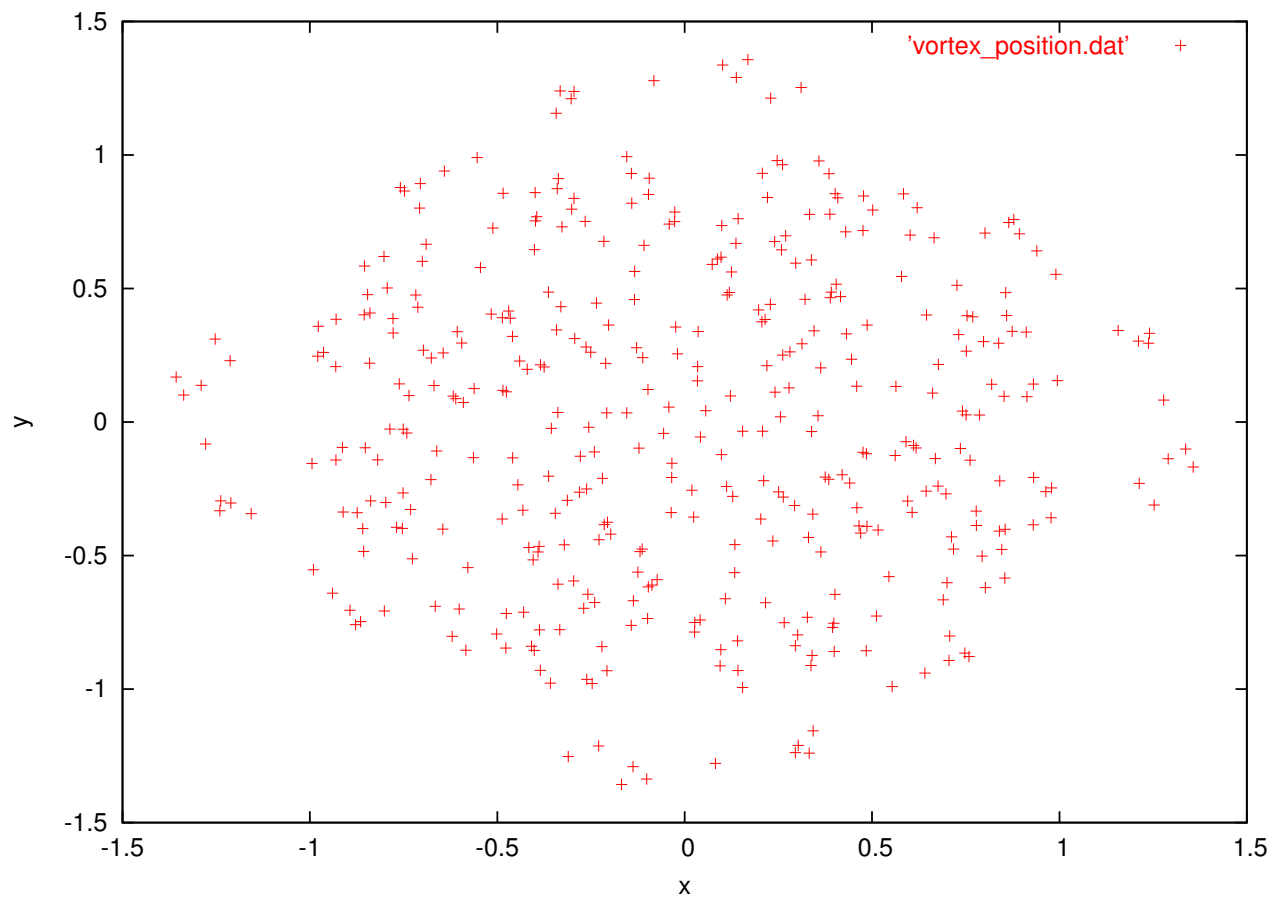


Figure 4.46: $t=0$

Figure 4.47: $t=1$

Figure 4.48: $t=2$

Figure 4.49: $t=3$

Figures 4.46-4.49 show the outcome of the computation at 4 different time steps. The plot above on the left shows the initial grid on which the vortices are arranged. The plots at $t=1, 2$ and 3 show the distortion of the grid after some time has elapsed. If the equations are to be solved analytically, the particles would move following circular trajectories of different radius. As confirmed in the results of Cottet, this does not happen because of the gradient of vorticity that tends to deflect the particles trajectories. The result is that the particles spiral outward, as proved by Cottet ([50]) and Lewis ([46]). The accuracy of the Runge Kutta scheme decreases rapidly as the particles move from their initial position. This is due to the distortion of the grid after a few time steps. As it has been pointed out in the introduction of the vortex methods, the condition of particles overlapping is essential and has to be maintained for the convection to be as accurate as possible. Therefore the use of an interpolation scheme is strongly recommended after a few time steps, in order to reinstate the initial regularity of the grid.

After having investigated the advantages and drawbacks of the vortex particle method for unbounded domains, it is time to show the results that have been achieved applying it to bounded domains. The steps shown in Lewis ([46]) have been closely followed in the present dissertation and the method has been applied for the solution of a uniform flow past a wedge-shaped body. It is widely accepted that the flow past a wedge-shaped body is easier to solve than the flow past a bluff body. For this reason our discussion will examine the flow

past the wedge. In the case of the shape shown in figure 4.50, it can be asserted with a good level of confidence that the flow will separate at edges of the vertical side. The strength of the vortices is established as described in [46]. Moreover, the direction and the gap between the separation point and the vortex needs to be carefully considered in order for the solution not to blow up. As described in earlier sections, the closer the vortex to the panel, the larger the velocity induced. Therefore an appropriate value of the gap is chosen, as shown in figure 4.50.

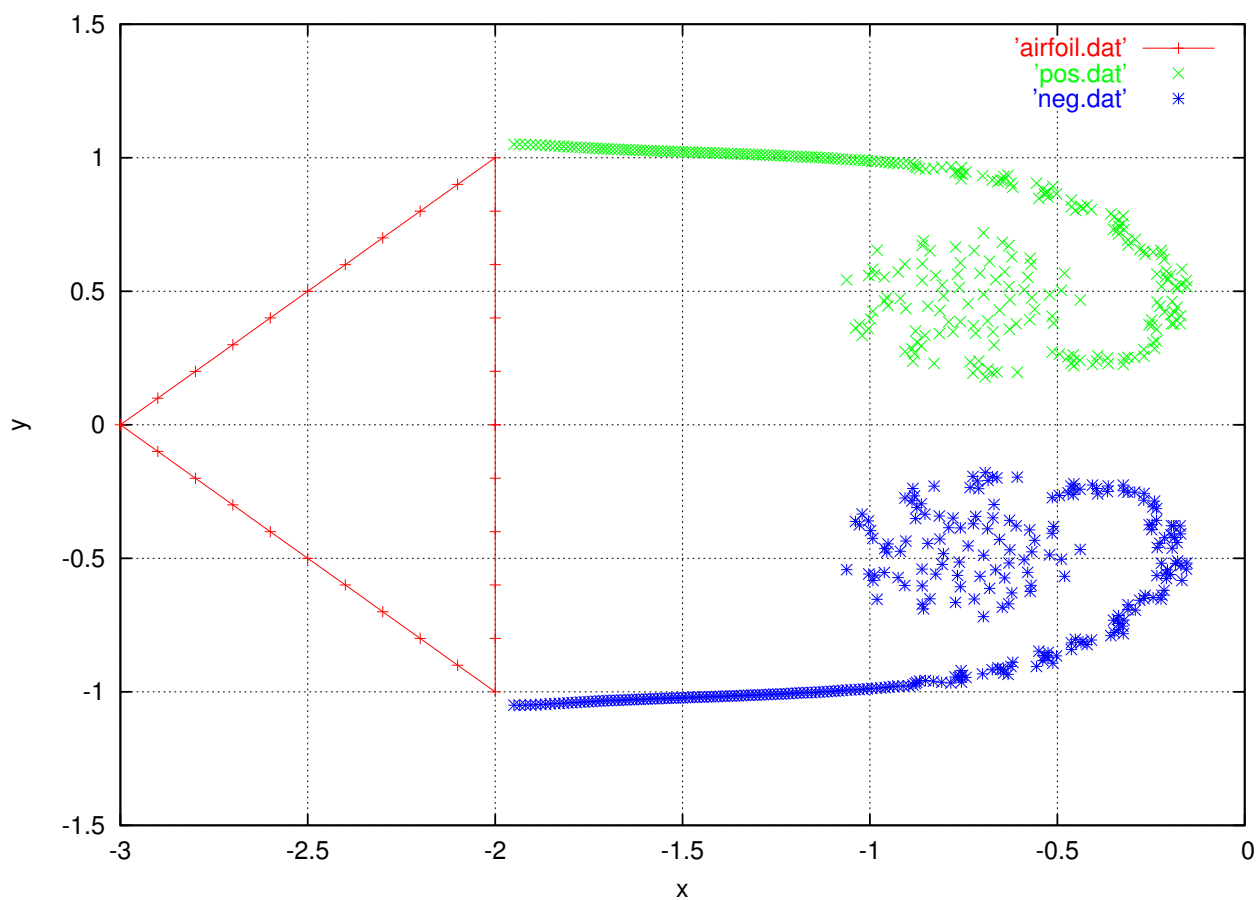


Figure 4.50: Results obtained with present VPM for flow past wedge with prescribed separation points

The comparison between figures 4.50 and 4.51 show a qualitative agreement between the two shedding models employed. In figure

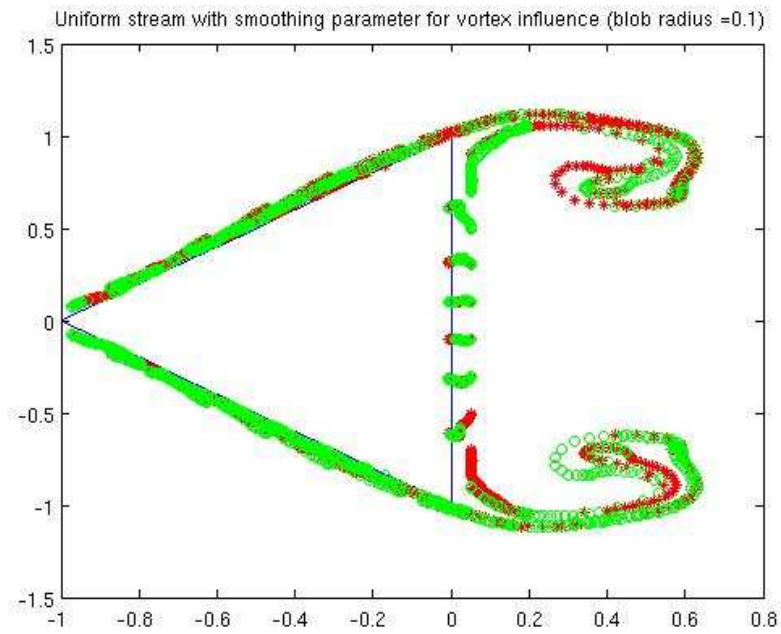


Figure 4.51: Results obtained with present VPM for flow past wedge without prescribed separation points

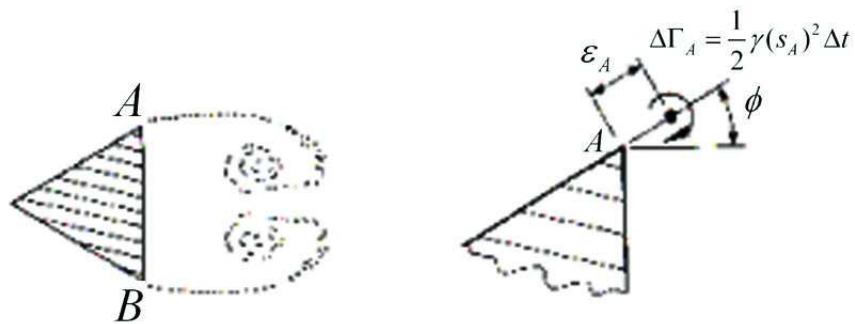


Figure 4.52: Comparison with figure 4.50, taken from [46]

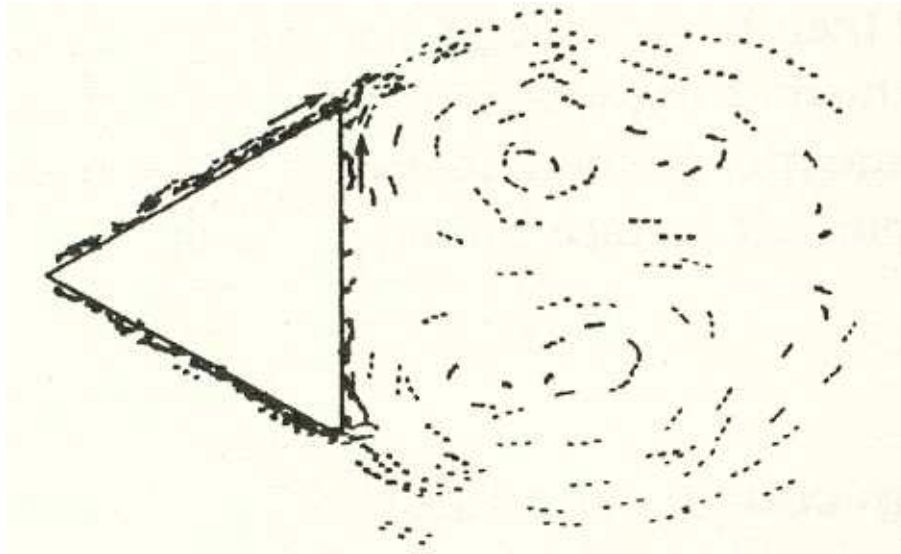


Figure 4.53: Comparison with figure 4.51,taken from [46]

4.51,the roll up of the wake seems to occur earlier than in figure 4.50,due to the interaction of the vortices coming from the diagonal sides with the ones released from the vertical side. The agreement with the results shown in 4.52 and 4.53 is also quite good,showing that the model implemented can predict the separation reasonably well as long as the body presents a sharp edge. This comparison has to be considered from a qualitative point of view only, as the assessment has not been taken further.

4.5 Aeroelasticity of Flapping Wings

The model shown in figure 3.15 has been used for the aeroelastic coupling. The origin of the coordinate system is taken to be the elastic axis. The two series of the parameters chosen for the computation are:

$$\begin{aligned}
 a = -0.4 \quad \mu_{AD} = 4 \quad \xi_\alpha = 0.2 \quad r_\alpha^2 = 0.25 \\
 a = -0.4 \quad \mu_{AD} = 2 \quad \xi_\alpha = 0.4 \quad r_\alpha^2 = 0.25
 \end{aligned}$$

For the meaning of the parameters the reader is referred to section 3.7. For each value of Ω the flutter speed of the airfoil has been sought analyzing the response of the wing. Considering the time that each computation takes and the little damping (or amplification) that the aerodynamics may provide, the time step used is $\Delta t = 0.1$. This is small enough for the computation since the adimensional form of the equations has been used. With this time step the tests have been run for enough time to assess whether instability arises.

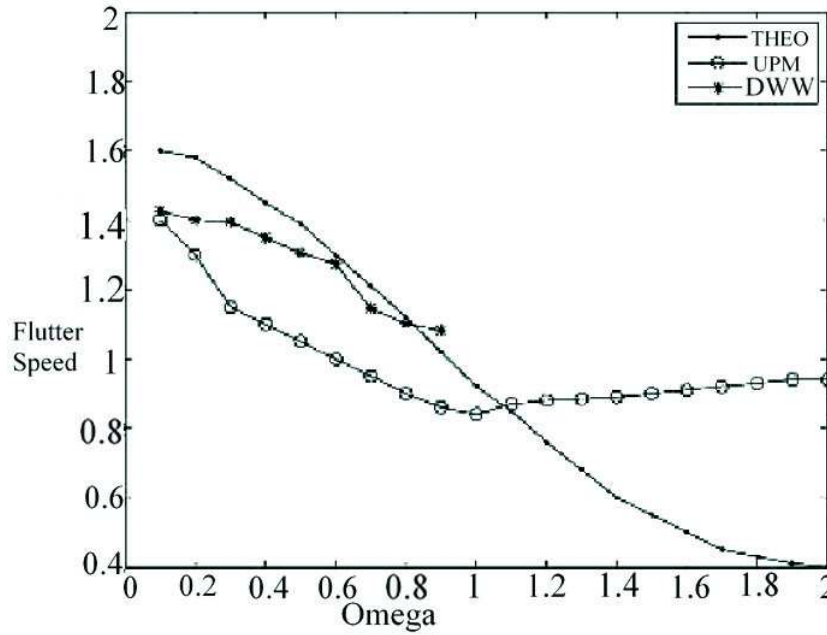


Figure 4.54: Flutter Analysis for series 1

Figures 4.54 and 4.55 represent the results obtained with the response analysis and they are compared with the ones found in Bisplinghoff[120]

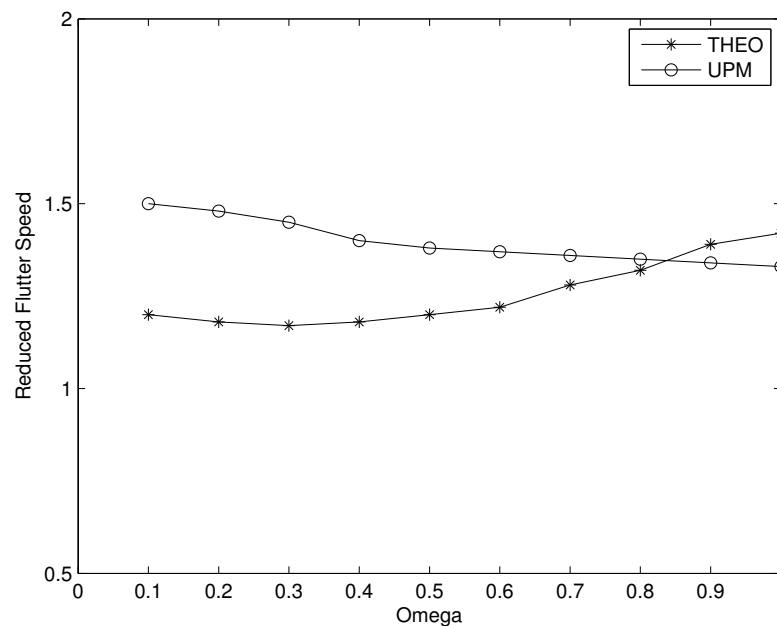


Figure 4.55: Flutter Analysis for series 2

(chapter 9-2, pp. 538-539). The deviation of the analysis made with UPM is not neglectable, and it is more evident in fig. 4.54 beyond $\Omega = 1$. After that frequency ratio, the linear theory predicts a drop in the flutter speed down to 0.4 when $\Omega = 2$, whereas UPM yields a flutter speed that is nearly constant. The agreement with the theoretical results is only qualitative.

Similar results are found for DWW, that is closer to the theoretical results in the range plotted. But it deviates by far from $\Omega = 1$ and the results beyond this value have not been plotted. For Ω greater than 1 the pitch frequency becomes lower than the plunge one, and UPM shows already that the flutter speed remains constant after that value.

The interesting aspect of the results is that the flutter speed predicted

with UPM is not constantly lower than the theoretical one. This should be due to the different assumptions on which the two methods are based on. When Ω starts going up, the natural frequencies of the airfoil increase and the frequency of oscillation of the airfoil changes. The main difference in the two methods that can cause the results to be so different is the wake, that is flat in Theodorsen, but free to move in UPM. The influence of the time step should be considered as well, as this may affect the calculation. Jones[3] shows that the step size affects the value of the flutter speed, but not the trend. This matter needs further investigation to be clarified.

A positive result is that UPM and Theodorsen predict stability of the airfoil for all the frequency ratios in case 1 when $\xi_\alpha = 0$. In this case the response of the wing to an initial perturbation is always damped out by the system.

The response of the system for different speeds is shown in figures 4.56 and 4.57. The speeds chosen for the computation are one below the flutter, one that shows a nearly constant amplitude, and a speed where flutter has already occurred. Due to the amount of time required for each computation, the velocity at which instability appears is not accurately computed, therefore the plots are subject to an error. The flutter speed has been appointed as that speed at which the amplitude of the response increases or decreases so slowly that its change is neglectable within the time window depicted.

The perturbation in the cases examined has been given in pitch, fix-

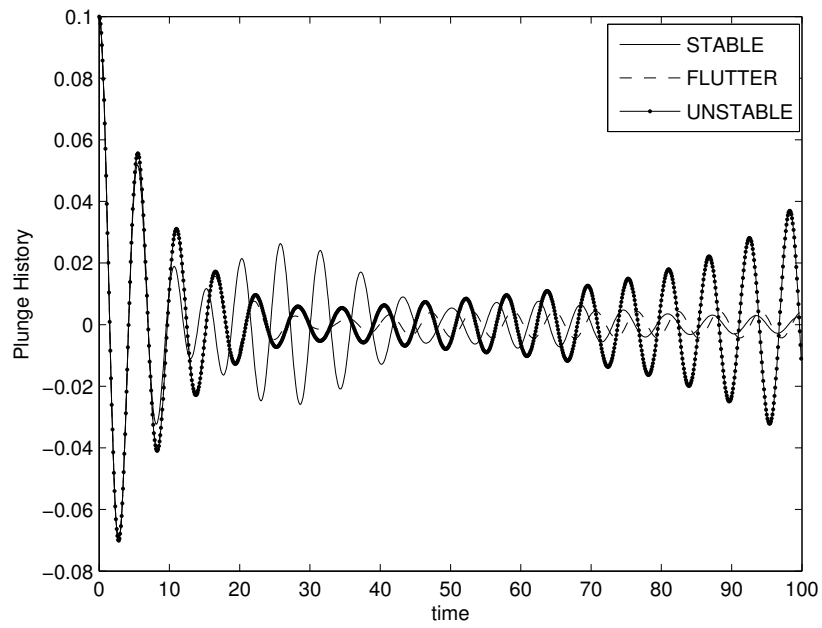


Figure 4.56: Plunge Response versus time

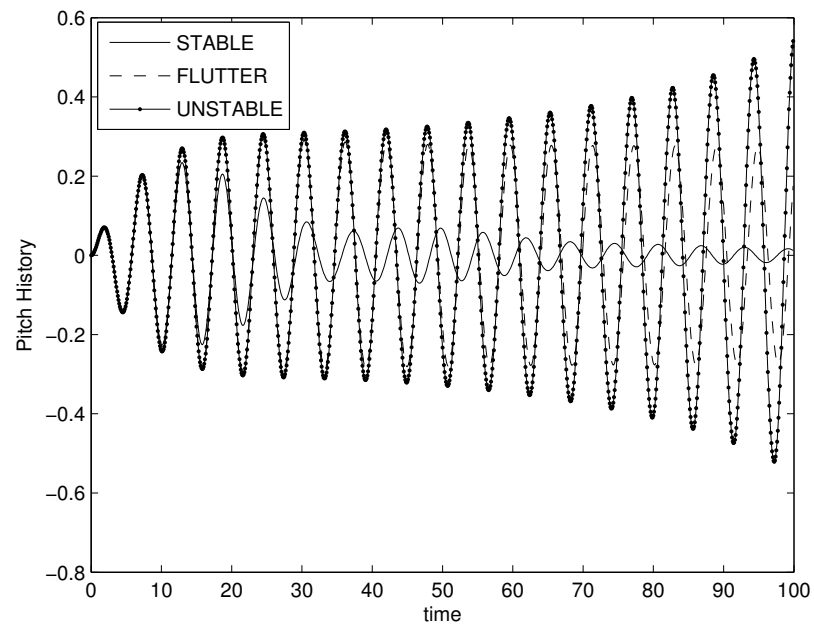


Figure 4.57: Pitch Response versus time

ing an initial angle of attack. Initial conditions can be given along the other degrees of freedom, but this should not change the flutter boundaries of the system. In the linear theory flutter occurs when one of the poles of the system crosses the imaginary axis towards the positive plane. This happens regardless of the amount of initial perturbation given to the system.

With UPM things are slightly different. It is still true that the system is stable every degree of freedom excited. Therefore the same can be said if more than one degree of freedom is excited at the same time. The aspect that must be taken care of is the amount of perturbation given. To excite the system with a big initial value of plunge/pitch would cause the circulation built around the airfoil to be so high that the wake panel released would affect too much the forces around the airfoil.

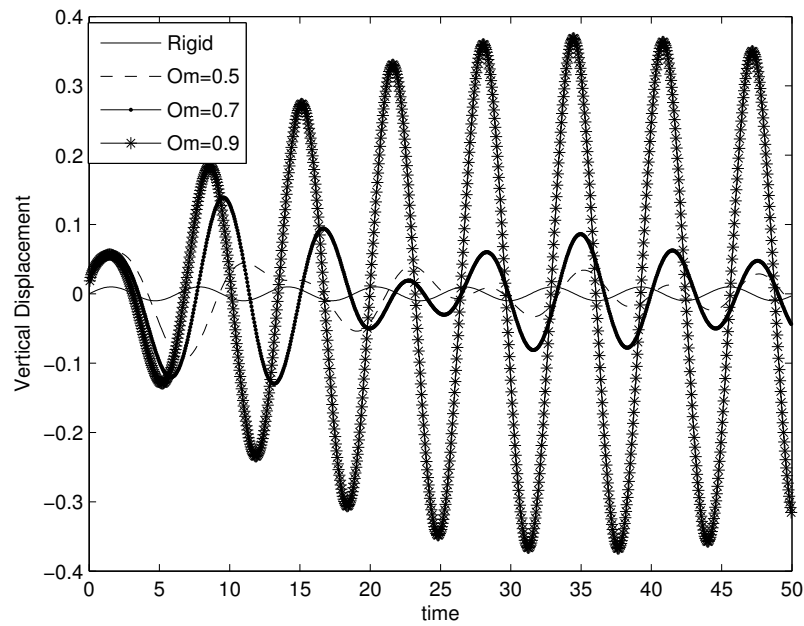
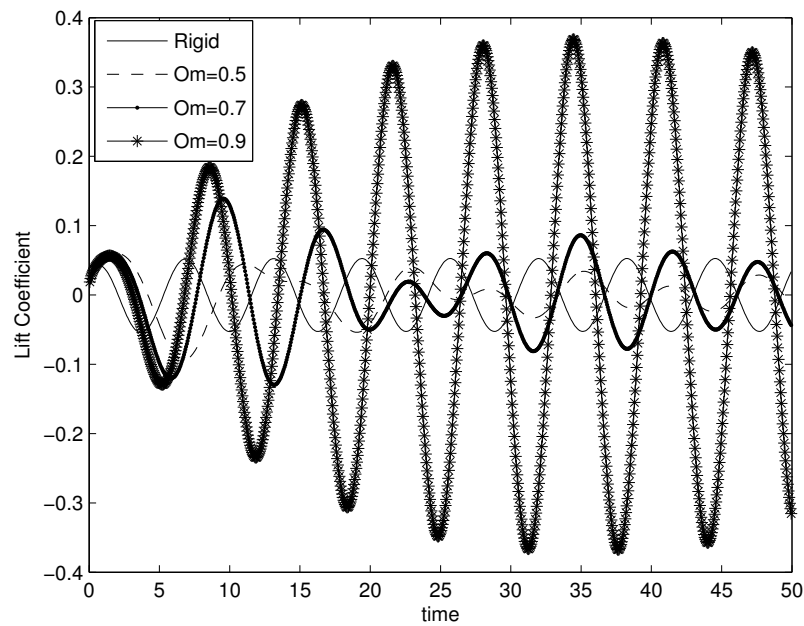
A big perturbation would not be acceptable not only because it would cause the system to explode, but also because the model would yield results that are not consistent with the assumptions made. Lagrange equations are valid under the assumption of small oscillation, so that in the expression of the kinetic energy $\cos \alpha$ and $\sin \alpha$ can be replaced respectively with 1 and α [3]. Furthermore, the panel method is not capable of predicting correctly the aerodynamic forces developed at very high angle of attack.

4.5.1 Aeroelastic Response of Flapping Wings

In this section the response is studied for a wing that is given a prescribed motion in time. Unlike the previous section, in which the perturbation was given only at the first time step and then the system was left to evolve on its own, in the present one the motion is continuous in time. In particular sinusoidal motions will be studied. This investigation is aimed at assessing whether there is a real advantage in considering the flexibility of the wing, not to establish quantitatively the amount of the advantage.

The freestream speed is within the flutter boundary found in the previous section: the system is stable and as long as the oscillations are small we are assured that flutter does not occur. The parameters are set to the following values: $U_\infty = 25m/s$, $\xi_\alpha = 0.2$ and $r_\alpha^2 = 0.25$. The amplitude of the motion has been set to 0.01, which is 1% of the chord. We want to study how the aerodynamics acts on the system when the frequency of oscillation approaches the resonance frequency. In a two-degree-of-freedom airfoil there are two resonance frequencies. In this study we want to assess the effect of the aeroelastic oscillations when only the plunge is excited. Therefore the torsional stiffness has been set to a very high value compared to the bending stiffness, so that the airfoil is infinitely rigid in pitch.

The parameter Ω is varied from 0.5 to 1.5, and the value $\Omega = 1$ corresponds to the frequency of the prescribed motion, when resonance occurs.

Figure 4.58: Plunge displacement versus Ω Figure 4.59: Lift Coefficient versus Ω

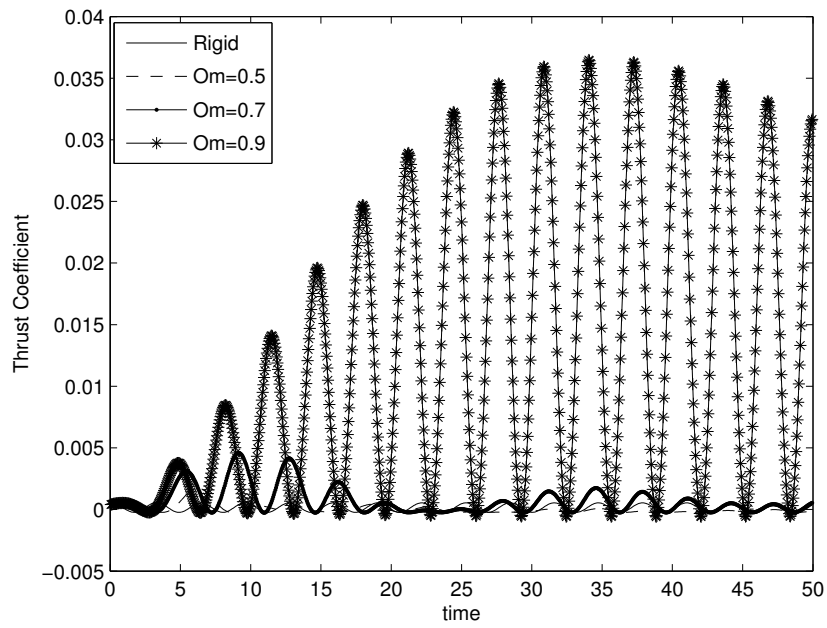
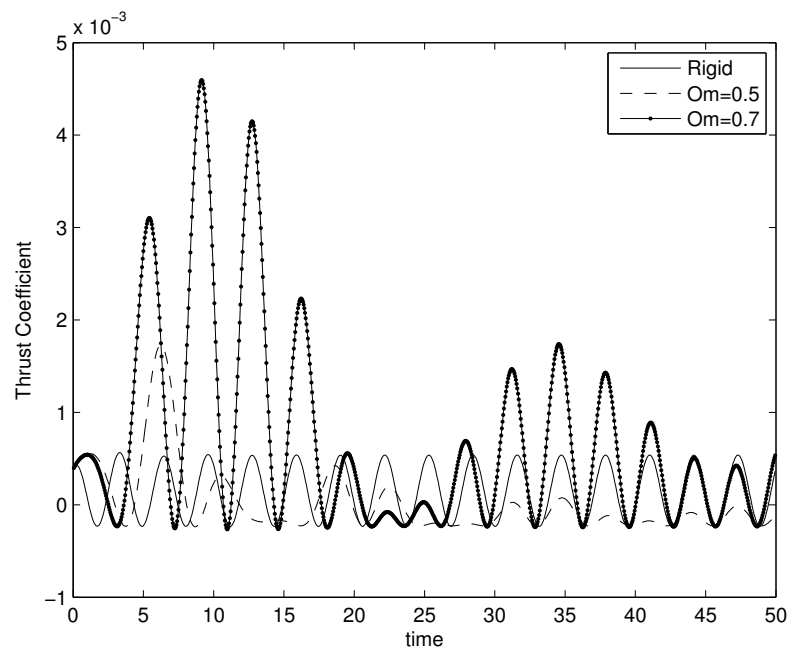
Figure 4.60: Thrust Coefficient versus Ω 

Figure 4.61: Zoom of the figure 4.60

Figures 4.58 to 4.61 show how the forces on the airfoil can be greatly increased approaching the resonance frequency of the structure. The system has no structural damping, and if the aerodynamics is switched off the amplitude of the motion would get much higher when $\Omega \rightarrow 1$. The simulation for the frequency of excitation equal to the resonance frequency have not been reported because the amplitude reached is too high compared with the others. The plots show that after the transient motion, in the tests with Ω far from 1 the response tends to return to the rigid value. The contribution that the flexibility of the wing gives in these cases is very small, even neglectable for $\Omega = 0.5$ and 1.5. The aeroelastic effect is also represented by the phase delay between the rigid and the elastic motions. Even in the cases where the amount of the elastic response is not significant, the phase delay shows the effect of the wing vibrating in the airflow. From the results just shown it can be concluded that the flexibility of the wing can increase the forces generated from the system. The reason of this happening is overall the inertia of the wing, whereas the aerodynamics does not have a significant contribution. In this case it can be said that the aerodynamics is beneficial in the sense that it damps the oscillations of the system, that would increase indefinitely if the only present damping had been structural. Therefore it can be stated that, although the aerodynamics does not directly increase the forces produced by the wing, it brings an advantage because the system can be excited at a higher frequency than with no aerodynamics. This effect can be clearly seen in experimental results shown in Marianetti [121].

Chapter 5

Conclusions and Recommendations

In this thesis the modelling of the aerodynamics around flapping wings for Micro Air Vehicles applications has been attempted. A numerical analysis of two dimensional flapping wings has been carried out with three methods. The aim of the investigation has been the development of a tool that could be used in the design phase of the aircraft for optimization.

The first part of the thesis has been dedicated to the development and validation of an unsteady panel method. To show the advantages of using an unsteady panel method over simplified linear methods, the former has been compared to Theodorsen and UTIAS, a method for flat plates featuring free wake convection. The improvement obtained switching from a flat wake - flat plate to deformable wake - thick airfoil, going through flat plate - deformable wake, has shown the importance

of modelling the flowfield correctly. Therefore a big effort has been made in order to shed the wake as accurately as possible.

One of the contributions of this investigation is the implementation of a new method to convect the wake elements after they have been released from the airfoil's trailing edge. The rearrangement of the equations has taken from a continuous wake, built up of doublets, to an equivalent discrete one, represented with point vortices. The validation of the results with the ones obtained with similar models has led to the conclusion that this wake model allows the flowfield and the aerodynamic forces to be represented correctly. The advantage of the panel method over the other models employed in the thesis is the more accurate way in which the airfoil and the wake are represented. This is reflected in a better agreement with the results found in literature. The advantage of using the convection scheme presented over other ones already in use in the literature is the fact that it is not necessary to introduce an additional point at the centre of the panels to concentrate the vorticity. The only points that are used are the vertexes of the panels. In terms of results the differences with the other schemes used for comparison (e.g.: Jones and Platzer) are small, hence we can conclude that the model employed can work very well to represent flows with small regions of separation.

A second contribution of the investigation is the analysis of complex configurations with two airfoils flapping closely to each other. While the *ground effect* configuration is of more practical interest but it does

not pose any challenge to the convection scheme used, the *Schmidt Propeller* arrangement has been used as an important test for the wake convection scheme. The results obtained with the two configurations have shown that the propulsive force and the efficiency can be greatly enhanced using two airfoils placed close enough to allow the wakes to exploit mutual interference. The study of alternative configurations has been deemed crucial given the importance that an increase in efficiency has for flapping wings. Furthermore, the analysis of two airfoils in a *Schmid Propeller* configuration has shown how well the wake is represented with the current model. The convection of wake elements drifting very close to the airfoils' contour has required a smaller time step to be resolved reasonably well, but it did not cause a great computational disadvantage.

A partial contribution of the thesis has been the adaptation of the unsteady panel method to the solution of more real flowfields, where separation occurs due to the high angles of attack achieved. The development of a vortex particle method has been attempted but not completed. Results have been obtained for bodies in which the separation points can be easily spotted, like a wedged shape. These have shown a qualitative agreement with the results the model has been validated against. The potential of vortex particle methods and their suitability for treating separated flows suggest that this road should be followed to improve the representation of the flowfield around flapping wings.

An important contribution of the thesis has been the study of flexible airfoils through integration of the potential flow models into a structural solver. The simplicity of the structure did not allow a deeper study of how a wing performs when the flexibility is switched on. Nevertheless the study has been fundamental to understand whether the forces produced by a flapping wings can be increased by building the wing with a more sophisticated structure. The analysis has been carried out on a simple model with two degrees of freedom. The structural parameters have been varied in order to assess their effect on the forces produced. The results have shown that the increase in the amplitude of oscillation caused by the inertial forces enhances the aerodynamic forces. Furthermore, the aerodynamics has proved to be beneficial because of the damping effect over the amplitude. The presence of the airflow could make it possible for the wing to oscillate at a frequency closer to resonance than without aerodynamics. Therefore a great advantage could be obtained taking into account the wing flexibility. This is a very important step towards the development of an optimized flapping wing.

References

- [1] F. Bohorquez D.J. Pines. Challenges facing future micro-air-vehicle development. *Journal of Aircraft*, 2006.
- [2] Lock G.D. Woods M.I., Henderson J.F. Energy requirements for the flight of micro air vehicles. *The Aeronautical Journal*, 2001. Paper No. 2546.
- [3] Platzer M.F. Jones K.D. Time-domain aeroelastic analysis of a two airfoil system with application to unsteady rotary wing flow-fields. In *AIAA 95-0337, 33rd Aerospace Sciences and Meeting Exhibit, Reno, NV*, 1995.
- [4] Platzer M.F. Jones K.D. Time-domain analysis of low speed flutter. *AIAA Journal*, 1996. Vol. 34, No. 5.
- [5] Platzer M.F. Jones K.D. Numerical computation of flapping-wing propulsion and power exctraction. In *AIAA-97-0826, 35th Aerospace Sciences Meeting and Exhibit, Reno, NV*.
- [6] Platzer M.F. Jones K.D. An experimental and numerical investigation of flapping-wing propulsion. In *AIAA 99-0995, 37th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV*, 1999.

- [7] Platzer M.F. Jones K.D. Flapping-wing propulsion for a micro air vehicle. In *AIAA-2000-0897, 38th Aerospace Sciences Meeting and Exhibit, Reno, NV*.
- [8] Platzer M.F. Jones K.D., Lund T.C. Experimental and computational investigation of flapping wing propulsion for micro air vehicles. In *bound in Fixed and Flapping Wing Aerodynamics for Micro Air Vehicle Applications, by Mueller J.T.*, 2001.
- [9] Platzer M.F. Jones K.D. On the use of vortex flows for the propulsion of micro-air and sea vehicles. In *Applied Vehicle Technology Panel (AVT) Meeting Week, Norway*, 2001.
- [10] Papadopoulos J. Platzer M.F. Jones K.D., Brashaw C.J. Development and flight testing of flapping wing propelled micro air vehicles. In *2nd AIAA Unmanned Unlimited Systems, Technologies and Operations (Aerospace, Land and Sea) Conference and Workshop San Diego, (CA)*. 2003.
- [11] Papadopoulos J. Platzer M.F. Jones K.D., Brashaw C.J. Bio-inspired design of flapping-wing micro air vehicles. *The Aeronautical Journal*, 2005. Vol.109, No. 108.
- [12] Allen Plotkin Joseph Katz. *Low Speed Aerodynamics*. Cambridge University Press, 2001.
- [13] Center K.B. Jones K.D. Numerical wake visualization for airfoils undergoing forced and aeroelastics motions. In *AIAA 96-0055*,

- 34th *Aerospace Sciences Meeting and Exhibit, Reno,Nevada*, 1996.
- [14] M.F. Platzer Jones K.D., C.M. Dohring. Wake structures behind plunging airfoils:a comparison of numerical and experimental results. In *AIAA 96-0078, 34th Aerospace Sciences Meeting and Exhibit, Reno,Nevada*, 1996.
- [15] Platzer M.F. Jones K.D., S. Davis. Oscillating-wing power generation. In *Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference,San Francisco,CA*, 1999.
- [16] Platzer M.F. Jones K.D. The unsteady aerodynamics of flapping-foil propellers.
- [17] H.Tuncer Platzer M.F. Jones K.D., J.C.S. Lai. Computational and experimental investigation of flapping-foil propulsion. In *ISABMEC ,Tokai University Pacific Center, Honolulu,Hawaii,USA*, 2000.
- [18] Platzer M.F. Jones K.D. On the prediction of dynamic stall onset on airfoils in low speed flow. 2001.
- [19] Platzer M.F. Jones K.D., Duggan S.J. Flapping-wing propulsion for a micro air vehicle. In *AIAA 2001-0126, 39th Aerospace Sciences Meeting and Exhibit, Reno,Nevada*, 2001.
- [20] Platzer M.F. Jones K.D., K. Lindsey. An investigation of the fluid-structure interaction in an oscillating-wing micro-hydropower generator.

- [21] Brashaw C.J. Papadopoulos J. Platzer M.F. Jones K.D., M. Nakashima. On flow separation control by means of flapping wings.
- [22] Jones K.D. Platzer M.F. Flapping wing aerodynamics - progress and challenges. In *AIAA 2006-500, 44th Aerospace Sciences Meeting and Exhibit, Reno (NV)*, 2006.
- [23] Jones K.D. Platzer M.F. On the design of efficient micro air vehicles. In *First International Conference Design and Nature, Udine, Italy*, 2002.
- [24] J.D. DeLaurier R.F. Larijani. A nonlinear aeroelastic model for the study of flapping wing flight. In *Fixed and Flapping Wing Aerodynamics for Micro Air Vehicle Applications*, volume 195 of *LNCS*, pages 399–428, Reston,VA, 2001. America Institute for Aeronautics and Astronautics.
- [25] J.M.V. Rayner. A vortex theory of animal flight.part 1. the vortex wake of a hovering animal. *Journal of Fluid Mechanics* (1979),vol. 91,part 4,pp.697-730.
- [26] J.M.V. Rayner. A vortex theory of animal flight.part 2. the forward flight of birds. *Journal of Fluid Mechanics* (1979),vol. 91,part 4,pp.731-763.
- [27] G.R. Spedding, A. Hedenstrom, and M. Rosen. Quantitative studies of the wakes of freely flying birds in a low-turbulence wind tunnel. *Experiments in Fluids*, 34(2):291–303, 2003.

- [28] N.H. Pratt P.J. Phlips, R.A. East. An unsteady lifting line theory of flapping wings with application to the forward flight of birds. *Journal of Fluid Mechanics* (1981), vol.112, pp.97-125.
- [29] Sir J. Lighthill. The inaugural goldstein memorial lecture-some challenging new applications for basic mathematical methods in the mechanics of fluids that were originally pursued with aeronautical aims. *The Aeronautical Journal*, Paper No.1755, 11 October 1989.
- [30] Salman A. Ansari. *A Nonlinear, Unsteady, Aerodynamic Model for Insect-like Flapping Wings in the Hover with Micro Air Vehicle Applications*. PhD thesis, Department of Aerospace, Power and Sensors, Cranfield University, RMCS, Shrivenham, 2004.
- [31] Willmott A. P. Thomas A.L.R. Ellington C.P., Van Den Berg C. Leading edge vortices in insect flight. *NATURE* Vol. 384, 19/26 Dec 1996.
- [32] Schmidt W. Der wellpropeller, ein neuer antrieb fuer wasser-, land- und luft-fahrzeuge. *Zeitschrift fur Flugwissenschaften*, Vol.13, 1965, pp. 472-479.
- [33] R. Zbikowski. On aerodynamic modelling of an insect-like flapping wing in hover for micro air vehicles. *Phil. Trans. R. Soc. London A*. (2002) 360, pp.273-290.
- [34] Theodor Theodorsen. General theory of aerodynamic instability and the mechanism of flutter. *NACA TR 496*, 1935.

- [35] C.Y. Chow A.M. Kuethe. *Foundations of Aerodynamics:Bases of Aerodynamic Design*. Cambridge University Press, 1977.
- [36] I.E. Garrick. Propulsion of a flapping and oscillating airfoil. *NACA Report 567*, 1936.
- [37] I.E. Garrick T. Theodorsen. Mechanism of flutter:a theoretical and experimental investigation of the flutter problem. NACA Report No. 685.
- [38] Knoller R. Die gesetze des luftwiderstandes. Flug- und Motortechnik (Wien), 3, No.21, pp. 1-7, 1909.
- [39] Betz A. Ein beitrag zur erklarung des segelfluges. Zeitschrift fur Flugtechnik und Motorluftschiffahrt, 3, pp. 269-272, Jan. 1912 (Wien), 3, No.21, pp. 1-7, 1909.
- [40] DeLaurier J. D. Fairgrieve J.D. Propulsive performance of two-dimensional thin airfoils undergoing large-amplitude pitch and plunge oscillations. *UTIAS Technical Note No. 226, CN ISSN 0082-5263*, 1982.
- [41] J.D. DeLaurier. An aerodynamic model for flapping-wing flight. The Aeronautical Journal, April 1993.
- [42] Di Scioscio Fabiana. Aeroelastic modelling of a micro flapping wing. Master Thesis, University La Sapienza, Rome, May 2007.
- [43] Luigi Morino Franco Mastroddi. Introduction to theoretical aeroelasticity for the aircraft. Notes of Aeroelasticity Course, University La Sapienza, Rome, 2003.

- [44] Hancock C.J. Basu B.C. The unsteady motion of a two-dimensional airfoil in incompressible inviscid flow. *Journal of Fluid Mechanics*, 1978.
- [45] Smith A.M.O. Hess J.L. Calculation of potential flow about arbitrary bodies. *Progress In Aeronautical Sciences*, 1966.
- [46] R.I. Lewis. *Vortex Element Methods For Fluid Dynamic Analysis Of Engineering Systems*. Cambridge University Press (United States).
- [47] Maskell E.C. On the kutta-joukowski condition in two-dimensional unsteady flow. TECH. MEMO AERO 1451, Royal Aircraft Establishment, Sept. 1972.
- [48] Victor V. Sychev G.L. Korolev Vladimir V. Sychev, A.I. Ruban. *Asymptotic Theory of Separated Flows*. Cambridge University Press, 1997.
- [49] Milazzo A. Davi G., Marretta R.M.A. Explicit kutta condition for unsteady two-dimensional high-order potential boundary element method. *AIAA Journal*, Vol. 35 No. 6 pp. 1080-1081, June 1997.
- [50] Petros D. Koumoutsakos Georges-Henri Cottet. *Vortex Methods: Theory and Practice*. Cambridge University Press, 2000.
- [51] K.V. Rao W.R. Van Dalsem, J.L. Steger. Some experiences with the viscous-inviscid interaction approach. NASA Technical Memorandum 100015, 1987.

- [52] M. Gennaretti L. Morino, F. Salvatore. A new velocity decomposition for viscous flows: Lighthill equivalent-source method revisited. *Comput. Methods Appl. Mech. Eng.*, 1999. 173.
- [53] A.J. Chorin. Numerical study of slightly viscous flow. *Journal of Fluid Mechanics* (1973),vol.57,part 4,pp.785-796.
- [54] M.A. Jones. The separated flow of an inviscid fluid around a moving flat plate. *Journal of Fluid Mechanics*, 2003.
- [55] G.H. Cottet. A new approach for the analysis of vortex flows in two and three dimensions. *Annales de l' I.H.P.*,section C,tome 5, no. 3 (1988),pp.227-285.
- [56] F. Pepin P. Koumoutsakos, A. Leonard. Boundary conditions for viscous vortex methods. *Journal of Computational Physics*,113,pp.52-61,1994.
- [57] M.S. Warren A. Leonard B. Jodoin G.S. Winckelmans, J.K. Salmon. Application of fast parallel and sequential tree codes to computing three-dimensional flows with the vortex element and boundary element methods. *ESAIM Proceedings*,Vol.1, 1996, pp.225-240.
- [58] A. Leonard. Vortex methods for flow simulation.
- [59] Jeff Eldredge. *A Dilatating Vortex Particle Method for Compressible Flow with Application to Aeroacoustics*. PhD thesis, California Institute of Technology, 2002.

- [60] R. Krasny. Desingularization of periodic vortex sheet roll-up. *Journal of Computational Physics*, 1986. 65.
- [61] Guido Morgenthal. *Aerodynamic Analysis of Structures Using High-Resolution Vortex Particle Methods*. PhD thesis, University of Cambridge, 2002.
- [62] S. Mas-Gallic P. Degond. The weighted particle method for convection-diffusion equations, part 1: The case of an isotropic diffusion. *Mathematics of Computation*, 1989. 53.
- [63] S. Mas-Gallic P. Degond. The weighted particle method for convection-diffusion equations, part 2: The anisotropic case. *Mathematics of Computation*, 1989. 53.
- [64] A. Leonard J. Eldredge, T. Colonius. A vortex particle method for compressible flows. AIAA 2001-2641, 15th AIAA Computational Fluid Dynamics Conference, Anaheim, California.
- [65] J.D. Eldredge. Efficient tools for the simulation of flapping wing flows. AIAA 2005-85, 43rd Aerospace Sciences Meeting and Exhibit, 10-13 January 2005, Reno Nevada.
- [66] J.D. Eldredge. The acoustics of two-dimensional leapfrogging vortices. AIAA 2005-2954, 11th AIAA/CEAS Aeroacoustics Conference, 23-25 May 2005, Monterey, California.
- [67] J.D. Eldredge. Numerical simulation of the fluid dynamics of 2d rigid body motions with the vortex particle method. *Journal of Computational Physics*, 2006.

- [68] J.D. Eldredge J. Toomey. Numerical and experimental investigation of the role of flexibility in flapping wing flight. AIAA 2006-3211, 36th Fluid Dynamics Conference and Exhibit, 5-8 June 2006, San Francisco, California.
- [69] A. Leonard J. Eldredge, T. Colonius. A vortex particle method for two-dimensional compressible flows. *Journal of Computational Physics*, 2002. 179.
- [70] A. Leonard J. Eldredge, T. Colonius. A general deterministic treatment of derivatives in particle methods. *Journal of Computational Physics*, 2002. 180.
- [71] J. Eldredge. The dynamics and acoustics of viscous two-dimensional leapfrogging vortices. *Journal of Sound and Vibration*, 2007. 301.
- [72] Lorena Barba. *Vortex Methods for Computing High Reynolds Number Flows: Increased Accuracy with a Fully Meshless Formulation*. PhD thesis, California Institute of Technology, 2004.
- [73] C.B. Allen L.A. Barba, A. Leonard. Vortex method with fully mesh-less implementation for high reynolds number flow computations.
- [74] G.S. Winckelmans P. Ploumhans. Vortex methods for high resolution simulations of viscous flow past bluff bodies of general geometry. *Journal of Computational Physics*, 2000.

- [75] Petros Koumoutsakos. Multiscale flow simulations using particles. *Annu. Rev. Fluid Mech.*, 2005. 37.
- [76] Zbikowski R. Szmelter J. Progress in aerodynamic studies of micro air vehicles based on insect-like flapping wings. AIAA 2003-419, 41st Aerospace Sciences Meeting and Exhibit, January 6-9 2003,Reno,NV.
- [77] M. Murayama K. Nakahashi T. Kato F. Togashi, Y. Ito. Flow simulation of flapping wings of an insect using overset unstructured grid. AIAA 2001-2619,15th Computational Fluid Dynamics Conference, 11-14 June 2001,Anaheim,California.
- [78] M. Raffel J. Kompenhans P. Wernert, W. Geissler. Experimental and numerical investigations of dynamic stall on a pitching airfoil. AIAA Journal,Vol. 34,No. 5,May 1996.
- [79] R. Foch D. Cylinder R. Ramarurti W. Sandberg J.Gardner D. Srull G. Piper P.Vaiana A. Kahn J. Kellogg, C. Bovais. Development and testing of unconventional micro air vehicle configurations. AIAA 2003-6656,2nd Unmanned Unlimited Systems,Technologies and Operations-Aerospace,15-18 September 2003,San Diego,California.
- [80] P.Vaiana J.Kellogg D. Cylinder R. Ramarurti, W. Sandberg. Computational fluid dynamics study of unconventional air vehicle configurations. The Aeronautical Journal, Vol.109,Number 1097,July 2005.

- [81] R. Lohner R. Ramarurti, W. Sandberg. Simulation of flow about flapping airfoils using a finite element incompressible flow solver. AIAA 99-0652, 37th Aerospace Sciences Meeting and Exhibit, January 11-14 1999, Reno, Nevada.
- [82] R. Lohner R. Ramarurti, W. Sandberg. Simulation of the dynamics of micro air vehicles. AIAA 2000-0896, 38th Aerospace Sciences Meeting and Exhibit, January 10-13 2000, Reno, Nevada.
- [83] Paul Young. *Numerical Simulation of the Unsteady Aerodynamics of Flapping Airfoils*. PhD thesis, School of Aerospace, Civil and Mechanical Engineering, The University of South Wales, Australian Defence Force Academy, 2005.
- [84] M. Kaya I. H. Tuncer. Optmization of flapping airfoils for maximum thrust. AIAA 2003-420, 41st Aerospace Sciences Meeting and Exhibit 6-9 January 2003, Reno, Nevada.
- [85] M.F. Platzer I.H. Tuncer. A computational study of flow separation characteristics and wake profiles behind a flapping airfoil. AIAA 99-0648, 37th Aerospace Sciences Meeting and Exhibit, January 11-14, 1999, Reno, Nevada.
- [86] M.F. Platzer I.H. Tuncer, J.C.S. Lai. A computational study of flow reattachment over a stationary/flapping airfoil combination in tandem. AIAA 98-0109, 36th Aerospace Sciences Meeting and Exhibit, January 12-15 1998, Reno, Nevada.

- [87] M.A. Ortiz M.F. Platzer I.H. Tuncer, J.C.S. Lai. Unsteady aerodynamics of stationary/flapping airfoil combination in tandem. AIAA 97-0659, 35th Aerospace Sciences Meeting and Exhibit, January 6-10 1997,Reno,Nevada.
- [88] C.M. Wang I.H. Tuncer, J.C Wu. Theoretical and numerical studies of oscillating airfoils. AIAA Journal, Vol.28,No.9, September 1990.
- [89] Neace K.S. Platzer M.F. Aerodynamic analysis of flapping wing propulsion. AIAA 93-0484, 31st Aerospace Sciences Meeting and Exhibit, January 11-14 1993,Reno,NV.
- [90] H.H.Chen T. Cebeci, E. Besnard. Calculation of multielement airfoil flows,including flap wells. AIAA 96-0056, 34th Aerospace Sciences Meeting and Exhibit, January 15-18 1996,Reno,Nevada.
- [91] P.O. Persson M. Drela J. Peraire S.M. Swartz K.S. Breuer D.J. Willis, E.R. Israeli. A computational framework for fluid structure interaction for biologically inspired flapping flight.
- [92] K.S. Breuer D.J. Willis, J. Peraire. A computational investigation of bio-inspired formation flight and ground effect.
- [93] N.V. Kokshayski. Tracing the wake of a flying bird. Nature, vol.279, No.5709,pp.146-148.
- [94] C.P. Ellington C. Van Den Berg. The vortex wake of a hovering model hawkmoth. Phil. Trans. R. Soc. London B. (1997) 352, pp.317-328.

- [95] A.L.R. Thomas A.P. Willmott, C.P. Ellington. Flow visualization and unsteady aerodynamics in the flight of the hawkmoth, *manduca sexta*. Phil. Trans. R. Soc. London B. (1997) 352, pp.303-316.
- [96] C.P. Ellington C. Van Den Berg. The three dimensional leading-edge vortex of a hovering model hawkmoth. Phil. Trans. R. Soc. London B. (1997) 352, pp.329-340.
- [97] A.P. Willmott A.L.R. Thomas C.P. Ellington, C. Van Den Berg. Leading-edge vortices in insect flight.
- [98] K.G. Gotz M.H. Dickinson. The wake dynamics and flight forces of the fruit fly *drosophila melanogaster*. The Journal of Experimental Biology, 199, pp.2085-2104, 1996.
- [99] K.G. Gotz M.H. Dickinson. Unsteady aerodynamic performance of model wings at low reynolds number. The Journal of Experimental Biology, 174, pp.45-64, 1993.
- [100] J.J. Hu S.Y. Lin. Numerical study of flapping wing.
- [101] Banerjee J.R. Guo S. and Cheung C.W. The effect of laminate lay-up on the flutter speed of composite wings. *Journal of Aerospace Eng.*, 2003. Proc. of IMechE, Part G, Vol. 217, No G3, pp.115-122, ISSN 0954-4100, Professional Engineering Publishing.
- [102] Cheng W. Guo S. and Cui D. Aeroelastic tailoring of composite wing structures by laminate lay-up optimization. *AIAA Journal*.

- [103] Katz J. Vest M.S. Aerodynamic study of a flapping-wing-micro-uav. In *AIAA-99-0994, 37th Aerospace Sciences Meeting and Exhibit, Reno, NV*, 1999.
- [104] Gursul I. Heathcote S., Martin D. Flexible flapping wing propulsion at zero freestream velocity. In *AIAA-2003/3446, 33rd AIAA Fluid Dynamics Conference, Orlando, FL*, 2003.
- [105] Taylor R.L. Hilber H.M., Hughes T.J.R. Improved numerical dissipation for time integration algorithms in structural dynamics. *Earthquake Engineering and Structural Dynamics*, 1977. Vol.5,pp.283-292.
- [106] Genna F. Bardella L. Newmark's time integration method from the discretization of extended functionals. *Journal Of Applied Mechanics*, 2005. Vol.72,pp.527-537.
- [107] Smith M.J.C. The effect of flexibility on the aerodynamics of moth wings: Towards the development of flapping-wing technology. In *AIAA-1995-0743, 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV*, 1995.
- [108] Michael J.C. Smith. *Simulating Flapping Insect Wings Using an Aerodynamic Panel Method: Towards the Development of Flapping Wing Technology*. PhD thesis, Purdue University, 1995.
- [109] v. Baskar A. Muniappan, V. Duriyanandhan. Lift characteristics of the flapping wing micro air vehicle. AIAA 2004-6331,

- 3rd Unmanned Unlimited Technical Conference, Workshop and Exhibit, 20-23 September 2004, Chicago, Illinois.
- [110] v. Baskar A. Muniappan, V. Duriyanandhan. Lift and thrust characteristics of flapping wing micro air vehicle (mav). AIAA 2005-1055, 43rd Aerospace Sciences Meeting and Exhibit, 10-13 January 2005, Reno, Nevada.
- [111] J.D. Eldredge R.K. Shukla. An inviscid model for vortex shedding from a deforming body. Theoretical and Computational Fluid Dynamics Manuscript.
- [112] W.A. Silva. *Discrete-Time Linear and Nonlinear Aerodynamic Impulse Responses for Efficient CFD Analysis*. PhD thesis, College of William and Mary, Williamsburg, VA, 1997.
- [113] Paolo Lisandrin. *Elements of Automated Aeroelastic Analysis in Aircraft Preliminary Design*. PhD thesis, TU Delft, 2007.
- [114] J. Khan. *Aerodynamic-Aerostructure Interaction of an Ultralight Aircraft*. PhD thesis, University of Sheffield, 2007.
- [115] Isogai K. Numerical study of transonic flutter of a two-dimensional airfoil. Technical Report of National Aerospace Laboratory, TR-617T, July 1980, Japan.
- [116] Spekrijse S.P. Hounjet M.H.L. Meijer J.J. Prananta B.B., Kok J.C. Simulation of limit cycle oscillation of fighter aircraft at moderate angle of attack. National Aerospace Laboratory, NLR-TP-2003-526, Amsterdam, The Netherlands.

- [117] Allegri G. Liani E., Guo S. Aeroelastic effect on flapping wing performance. In *AIAA 2007-2412, 48th AIAA Structural Dynamics and Materials Conference, Honolulu, USA*, 2007.
- [118] Allegri G. Liani E., Guo S. Aerodynamics and aeroelasticity of flexible flapping wings. In *International Forum for Aeroelasticity and Structural Dynamics, Stockholm, Sweden*, 2007.
- [119] Allegri G. Liani E., Guo S. Potential flow-based aerodynamic analysis of a flapping wing. In *AIAA-2007-4068, 25th Applied Aerodynamics Conference, Miami, USA*, 2007.
- [120] R.L. Halfman R.L. Bisplinghoff, H.Ashley. *Aeroelasticity*. Dover Publications, 1996.
- [121] Marianetti Cecilia. Rigid and elastic description of micro flapping wing motion for aeroelastic applications. Master Thesis, University La Sapienza, Rome, July 2007.