Adaptive Mesh Refinement and Simulations of Unsteady Delta-Wing Aerodynamics

Yann Le Moigne

KTH Aeronautical and Vehicle Engineering
Royal Institute of Technology
SE-100 44 Stockholm, Sweden

TRITA-AVE 2004:17
ISSN 1651-7660
ISBN 91-7283-770-5
To my mother and late father
Typtsätt i \LaTeX{}.

TRITA-AVE 2004:17
ISSN 1651-7060
ISBN 91-7283-770-5

Aksjomisk avhandling som med tillstånd av Kungliga Tekniska högskolan
fredagen den 4 juni 2004 kl. 10, sal E2, Lindsedtsvägen 3, Stockholm, för
teknisk doktorsgrads vinnande framlägges till offentlig granskning av
Yann Le Moigne.

© Yann Le Moigne 2004
Preface

The work presented in this doctoral thesis has been carried out between November 1999 and May 2004 mainly at KTH Aeronautical and Vehicle Engineering (Royal Institute of Technology) and partly at the Swedish Defence Research Agency (FOI). This work has been financed for a part by the Parallel and Scientific Computing Institute under project number 625933, "P-SCI-CFD National Code". This financing is greatly acknowledged.

Throughout these five years, from being a docile student at the beginning, I have learned to become an independent researcher. Completing this thesis improved my knowledge in a lot of fields like management, psychology, diplomacy, a lot in computer administration and eventually a bit in aerodynamics. Unfortunately, there is not enough material to write a thesis in each of these domains so only the last one is reported here. I also got to know a lot more about life in general and I strengthened my character through the ups and downs, both private and professional. So that at least that goal is reached ...

My supervisor Arthur Rizzi has to be thanked for introducing me to the world of research and giving me freedom in my work. I acknowledge his interest in my work the last year of my PhD. He has always been a source of motivation and encouragement all along the final stretch.

I am deeply indebted to Jesper Oppelstrup, my unofficial “second” supervisor, for his advice, always relevant questions and solution-oriented attitude.

Many people have to be thanked for their help with softwares, only a few are named here: first of all, Jan B. Vos, from CFS Engineering SA in Lausanne, for his availability to answer questions on NSMB; Peter Eliasson at FOI for the help with EDGE; Stephen Conway and Magnus Tormalm for their tips for using ICEM CFD and Pro Engineer.

I do not forget the researchers from the RTO AVT-080 and AVT-113 Task Groups. It has been a pleasant experience to meet all these (in my eyes) gods of delta-wing flow and I feel proud to have been accepted as one of them.

A special thanks goes to Stefan Görtz for sharing his experience of CFD tools with me. We have traded so many tips about ICEM CFD, EnSight, NSMB, EDGE, Latex and so on, that maybe this thesis would not have been ready in time without him. Our common view on our work has given matter to endless discussions and often made life a lot funnier too.

Finally, I would like to express my gratitude to my family and friends for their eternal support through the difficulties, despite the distance.


Yann Le Moigne
Abstract

This thesis deals with Computational Fluid Dynamics (CFD) simulations of the flow around delta wings at high angles of attack. These triangular wings, mainly used in military aircraft designs, experience the formation of two vortices on their lee-side at large angles of attack. The simulation of this vortical flow by solving the Navier-Stokes equations is the subject of this thesis. The purpose of the work is to improve the understanding of this flow and contribute to the design of such a wing by developing methods that enable more accurate and efficient CFD simulations.

Simulations of the formation, burst and disappearance of the vortices while the angle of attack is changing are presented. The structured flow solver NSMB has been used to get the time-dependent solutions of the flow. Both viscous and inviscid results of a 70°-swept delta wing pitching in an oscillatory motion are reported. The creation of the dynamic lift and the hysteresis observed in the history of the aerodynamic forces are well reproduced.

The second part of the thesis is focusing on automatic mesh refinement and its influence on simulations of the delta wing leading-edge vortices. All the simulations to assess the grid quality are inviscid computations performed with the unstructured flow solver EDGE.

A first study reports on the effects of refining the wake of the delta wing. A 70°-swept delta wing at a Mach number of 0.2 and an angle of attack of 27° where vortex breakdown is present above the wing, is used as testcase. The results show a strong dependence on the refinement, particularly the vortex breakdown position, which leads to the conclusion that the wake should be refined at least partly. Using this information, a grid for the wing in the wind tunnel is created in order to assess the influence of the tunnel walls.

Three sensors for automatic mesh refinement of vortical flows are presented. Two are based on flow variables (production of entropy and ratio of total pressures) while the third one requires an eigenvalue analysis of the tensor of the velocity gradients in order to capture the position of the vortices in the flow. These three vortex sensors are successfully used for the simulation of the same 70°delta wing at an angle of attack of 20°. A comparison of the sensors reveals the more local property of the third one based on the eigenvalue analysis. This latter technique is applied to the simulation of the wake of a delta wing at an angle of attack of 20°. The simulations on a highly refined mesh show that the vortex sheet shed from the trailing-edge rolls up into a vortex that interacts with the leading-edge vortex.

Finally the vortex-detection technique is used to refine the grid around a Saab Aerosystems Unmanned Combat Air Vehicle (UAV) configuration and its flight dynamics characteristics are investigated.

**Key words:** delta wing, high angle of attack, vortex, pitching, mesh refinement, UAV, vortex sensor, tensor of velocity gradients.
Sammanfattning

Avhandlingen handlar om CFD beräkningar runt deltagningar vid höga anfallsvinklar. Två virvlar bildas på ovansidan av triangelformiga vingar som används på militär flygplan. Avhandlingen är en simulering av virvelströmmningen genom att lösa Navier-Stokes ekvationer. Arbetets syfte är att förbättra förståelse av strömmningen och underlätta design av deltagningar genom utveckling av metoder som gör CFD beräkningar mer exakta och effektiva.


Avhandlingen andra del handlar om automatisk nätförfinning för CFD beräkningar runt deltagningar. I den delen, har den strukturerade strömningsslösern EDGE används och, enbart icke-viskösa beräkningar presenteras.


Tre numeriska sensorer för automatisk nätförfinning för simulering av virvelströmmningen presenteras i en senare studie. De två första sensorerna är baserade på strömningstvivelser (produktion av entropi och totalttryck) medan den tredje behöver en egenvärdeanalyse av hastighetsgradiententensor för att identifiera virvlar i strömmningen. De tre sensorerna ger bra resultat när de används för att beräkna strömmningen runt en deltagning vid anfallsvinkel 20°.

Den tredje sensorn används också för att räkna på en deltagningsvak. Simuleringar görs på ett väl förfinat nät och visar en interaktion mellan två virvlar, den som finns på vingens ovansida och en som bildas vid vingens baklant.

Slutligen används den sista nämnde sensorn för att förfinna ett nät runt en av Saab Unmanned Combat Air Vehicle konfigurationer och dessa flygdynamiska egenskaper studeras.
Dissertation

This doctoral thesis consists of a brief introduction to the research area, a short summary of the main results obtained and the following appended papers:

**Paper A**


**Paper B**


**Paper C**


**Paper D**


**Paper E**

Le Moigne, Y., “Simulations of delta-wing wake roll-up with mesh refinement”, 2004. To be submitted for publication.

**Paper F**

Division of work between authors

**Paper A**
Le Moigne performed the computations and wrote the paper. Rizzi supervised the work and contributed with valuable comments for the analysis of the results. Johansson brought an industrial point of view to the subject.

**Paper B**
Le Moigne performed the computations, implemented the necessary changes in the flow solver and wrote the paper. Rizzi supervised the work and contributed with valuable comments.

**Paper C**
Le Moigne created the numerical grids, performed the calculations and wrote the paper. Rizzi supervised the work and contributed with valuable comments.
Contents

Preface i

Abstract iii

Sammanfattning v

Dissertation vii

Division of work between authors ix

Introduction 1

1. Background ............................................... 1
2. Scope of the thesis ........................................ 5
3. Previous work in the field of research ...................... 6
   3.1. Pitching delta wings ............................... 6
   3.1.1. Experimental approach ........................... 6
   3.1.2. Numerical approach ............................. 8
3.2. Mesh refinement and vortex flows ...................... 9
4. Numerical methods ....................................... 11
   4.1. Mesh generation .................................... 11
   4.2. The structured flow solver NSMB ................. 12
      4.2.1. General description ........................... 12
      4.2.2. Moving grid approach ....................... 13
      4.2.3. Dual time stepping ........................... 14
   4.3. The unstructured flow solver EDGE ............. 14
      4.3.1. General description ........................... 14
      4.3.2. Automatic mesh refinement ................... 15
      4.3.3. Implementation of sensors for vortex flows .... 16
5. Summary of appended papers ............................ 17
   5.1. Paper A ........................................... 17
   5.2. Paper B ........................................... 18
   5.3. Paper C ........................................... 19
   5.4. Paper D ........................................... 20
   5.5. Paper E ........................................... 21
   5.6. Paper F ........................................... 22
6. Discussion ............................................. 23
7. Concluding remarks ...................................... 27
8. Future work ............................................ 27

References 31

Paper A A1-A11
<table>
<thead>
<tr>
<th>Paper</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Paper B</td>
<td>B1-B18</td>
</tr>
<tr>
<td>Paper C</td>
<td>C1-C25</td>
</tr>
<tr>
<td>Paper D</td>
<td>D1-D25</td>
</tr>
<tr>
<td>Paper E</td>
<td>E1-E18</td>
</tr>
<tr>
<td>Paper F</td>
<td>F1-F19</td>
</tr>
</tbody>
</table>
Introduction

1. Background

First developed in the 1930’s by German engineers and tested by the Americans in the 1950’s on the XF-92A test airplane, the delta wing concept (from the capital greek letter Δ) has made its way into operation from the Convair F-102A "Delta Dagger" to more modern fighters like Saab - BAE Systems Gripen (see Figure 1). Even if all the modern fighters do not have a delta wing, all of them exhibit the same aerodynamics with vortical flow emanating from strakes or chines at large angles of attack.

![Image of XF-92A and Gripen](image)

(a) XF-92A  
(b) Gripen

Figure 1: Examples of aircraft with delta wing planform

**Delta wing aerodynamics** The prime aerodynamic characteristic of delta wings is the formation of a pair of leading-edge vortices on the upper surface of the wing at an angle of attack (see Figure 2). The vortices are created by the rolling-up of the shear layer that separates at the leading-edge and is carried downstream by the longitudinal component of the free stream velocity. The rotating flow reattaches to the surface and can separate again to form a secondary vortex. Between the two primary vortices, the flow remains attached
to the wing. Note that Figure 2 is showing a very simplified flow field and that despite the numerous studies made on this type of flow, some aspects are not yet very well known. Three of them are for example transition from laminar to turbulent flow, both on the wing surface and in the rolling shear layer, the presence of vortex breakdown and the nonlinear interaction between vortices.

![Diagram of flowfield above a delta wing at angle of attack](image)

**Figure 2:** Schematic of the flowfield above a delta wing at angle of attack [1]

The two vortices are a source of energy with very high speeds and create a low surface pressure beneath them. It produces an additional lift force called “vortex lift” that increases until high angles of attack (around 35°). Compared to planes with rectangular planforms, this high stalling angle of attack is an advantage for maneuvering aircraft.

Unfortunately the lift does not increase indefinitely with the angle of attack. At large angles of attack the leading edge vortices experience an abrupt change with a sudden expansion of their section, an increase in dynamic pressure and a loss of axial velocity (see Figure 3). This phenomenon is called vortex breakdown or burst. The flow in the breakdown is very chaotic and turbulent although it still has a spiral behaviour. As the position of the breakdown moves upstream with an increasing angle of attack, this ultimately leads to the stall of the wing, i.e. maximum lift is attained and further increase leads to lower lift.

Although a lot of work has been done to characterize vortex breakdown, no unified theory has emerged to explain the phenomenon (see for example [3] for a presentation of the theoretical approaches). There is no general agreement either, to explain the difference (if any) between the two types of breakdown usually observed in experiment and shown simultaneously in Figure 3: the asymmetric spiral type (top) and the axisymmetric bubble type (bottom). As the phenomenon is not fully understood, the prediction of the position of vortex
breakdown relies only on experiments and numerical simulations. Work on simpler analytical methods to predict the breakdown position is still going on but there is not yet an approach that has been accepted unanimously.

**Maneuvering delta wings and dynamic lift.** NASA’s X-31 program has shown the feasibility of post-stall flight and the combat superiority of highly maneuverable aircraft [4]. This has lead to more research in the domain of maneuvering aerodynamics and in particular, maneuvering delta wings. Both experimental and numerical work has started on simple maneuvers: pitching or rolling wings. The major difference between a wing at a fixed attitude and one that is pitching is an hysteretic behaviour of the breakdown. By definition, hysteresis is a lagging or retardation of an effect. For delta wings, it means that the effect of the flow conditions (angle of attack, roll angle ...) on the vortex is not felt immediately but after some delay. Thus, depending on the history of the maneuver, even if the instantaneous flow conditions are the same, the flow field can be different. In the case of pitching delta wings, the vortex breakdown position depends not only on the current flow conditions (free stream velocity, angle of attack, ...) but also on the history of the wing motion i.e. increasing (upstroke) or decreasing (downstroke) angle of attack. This dynamic position of the vortex breakdown is also dependent on, among other parameters, the location of the axis of rotation, the amplitude of the motion (initial and final angles of attack) and the pitch rate, often given by

\[ k = \frac{\omega c}{2U_\infty} \]

for sinusoidal oscillations.

Figure 4 illustrates this phenomenon with experimental results of the dynamic vortex breakdown position for a pitching 70°-swept delta wing: in the upstroke

---

1In the literature, the definition \( k = \frac{\omega c}{U_\infty} \) is also used.
motion the position of the breakdown is delayed downstream of its static value (larger X/C) whereas in the downstroke motion, the breakdown remains closer to the apex of the wing (lower X/C). This hysteretic behavior of the breakdown creates an hysteretic loading of the wing with an overshoot of the static forces in the upstroke motion and an undershoot in the downstroke motion. In Figure 5, the advantage of a rapid pitch-up maneuver is obvious with the additional normal force obtained compared to the static case, the so-called “dynamic lift”.

![Figure 4: Dynamic vortex breakdown position vs. angle of attack [5]](image1)

![Figure 5: Static and dynamic normal force coefficient [5]](image2)

**UAVs and UCAVs** A new category of military aircraft is entering service in many air forces and more and more countries are working on the development of their own vehicles. These aircraft are Unmanned Aerial Vehicles (UAVs). They are designed for a large variety of missions but mainly for surveillance/reconnaissance purposes with loitering times much longer than manned aircraft. The most famous ones are General Atomics Aeronautical Systems “Predator” and Northrop Grumman “Global Hawk”. These designs are rather conventional with a fuselage and large aspect ratio wings. Newer generations can carry ordinances often in internal weapon bays and are called Unmanned Combat Air Vehicles (UCAVs). These vehicles are meant for high-risk missions where the presence of a pilot would be too costly, for example in the suppression of enemy air defences. Stealth requirements and vehicle maneuverability lead to more unconventional designs with low aspect ratio highly swept wings, V-tails and chines or strakes in front of the wing (see Figure 6). Although the configurations are not all with delta wings like the Northrop Grumman X47-A, their aerodynamics is dominated by vortices much like classical delta wings. The understanding and simulation of vortex flow has thus received a new in-
terest. Indeed, the prediction of an accurate flow field is vital for the design of these vehicles dominated by non-linear phenomena. The cross-coupled dynamics has to be understood and predicted in order to have an accurate flight dynamic model of the vehicle. Economic savings in both time and money can be realized if a simulation model can provide high-fidelity information about the aerodynamic characteristics that are necessary for designing a flight control system that is “first-time right” (see “New challenges in aircraft design” in [6]).

(a) Northrop Grumman X47-A  (b) Singapore Technologies MAV-1

Figure 6: Examples of UCAVs

This concludes the presentation of the background on vortex flow. An overview of the thesis is presented next, followed by a brief review of the work previously published in the domain of maneuvering delta wings and computational grid refinement for vortex flow simulations. Then comes the core of the thesis in the form of a summary of the appended papers. Finally, comments on the papers are made and potential future work presented.

2. Scope of the thesis

The work presented below and in the appended papers deals with numerical simulations of vortex flows, mainly around delta wings. The work can be divided into two related parts. First, time-accurate numerical simulations of a delta wing pitching to high angles of attack are presented. These simulations were performed with a mature structured flow solver. The aim of this study is to predict the vortex behavior above the delta wing in large amplitude maneuvers where the flow field is changing dramatically. In this first part, emphasis is put on unsteady effects and the prediction of the dynamic lift and general dynamic aerodynamics. In the second part, simulations of delta wing flows are performed on unstructured grids with a more modern but less mature flow solver. In this case, the focus is put on taking advantage of the unstructured approach to improve the accuracy of delta wing simulations, for steady state cases in a first step. It makes use of the more freedom of “grid design” that
the unstructured approach gives, in order to better capture details in the flow field and improve the prediction capability of the CFD tools. The work in this part is limited to steady simulations for which grid refinement techniques are developed and tested.

3. Previous work in the field of research

The domain of vortex flows as presented above is vast and has been the subject of decades of research. In order to put the present work into context, a short review of the work done in the field is presented. This is not meant to be an exhaustive review. Only the research related to the two main topics of this thesis, pitching delta wings and automatic mesh refinement, is presented.

3.1. Pitching delta wings

3.1.1. Experimental approach

A large variety of experiments has been carried out on delta wings and it is probably not wrong to say that all the wind or water tunnels in the world have seen models of delta wings being tested. These are not reviewed here. A number of review articles present the basic features of the flow around delta wing and list references, see for example [7]. The latest review articles [8,9] focus on the dynamic aspects of vortical flow, be them naturally unsteady or excited by wing motion. The general case of static wings and steady experiments is also reviewed in [8,9]. These two references although not specific to pitching delta wings give also a good overview of the subject. When it comes to pitching delta wings, the number of published results decreases but still remains important. Unfortunately for the CFD community, no experiment stands out with a complete set of data to be used for validation of simulations. Many different experimental set-ups are used be it in wind or water tunnels and even towing tanks, all having their pros and cons. It leads to experimental results focusing either on unsteady pressure measurements or forces measurements or flow visualization but never on all of these aspects at the same time.

Ashley et al. [10] review many of the works published until the late 80’s and present in more detail the results by Jarrah [11] together with numerical solutions. In [11], both flow visualization and force measurements on delta wings with different sweep angles are reported. Both sinusoidal pitching and ramp-and-hold2 motions are investigated at several reduced frequencies and for three different ranges of angles of attack.

The work by Soltani and Bragg cannot be omitted in a review of publications on pitching delta wings. It has already been shown in Figure 5 taken from [5] but [12] gives probably a better overview of their wind tunnel experiments:

---

2A motion where the angle of attack increases and is then kept constant at its maximum
pitching oscillations of a 70° delta wing from 0° to 55° at several reduced frequencies, Reynolds numbers and with sideslip that reveal the typical hysteresis loops in the history of the aerodynamic coefficients. Unfortunately, only forces are reported.

Reference [13], by Brandon from NASA Langley, serves as reference for the first part of the work presented in this thesis. In this paper, normal force coefficients are reported for sinusoidal oscillations with a constant semi-amplitude of 18° around several mean angles of attack at different reduced frequencies. Three wings are tested, one rectangular and two delta wings with sweep angles 45° and 70°, as well as a model of the F-18 fighter.

![Figure 7: Effect of the mean angle of attack on dynamic normal force characteristics [13]](image)

Rockwell et al. have performed experiments in a water tunnel more adapted for flow visualization [14] and Particle Image Velocimetry (PIV) measurements [15]. The flow visualization reveals different types of vortex breakdown depending on the frequency of the oscillations of the wing while the PIV measurements enable a detailed representation of the flow topology, for example in a cross flow plane above the wing.

Thompson, Barill and Nelson [16] have several references in which unsteady pressure distributions on pitching delta wings are reported. At a fixed location, the pressure also changes with hysteresis loops very similar to the integrated coefficients.

Myce et al. [17] present interesting results on a configuration study. They show that by modifying the rear part of a 76° delta wing, from a diamond to a double-delta wing, the motion of the vortex breakdown above the pitching wing is affected.
3.1.2. Numerical approach

When it comes to CFD simulations of pitching delta wings, one has to refer to the extensive work done by Visbal and Gordnier at Wright Patterson Air Force Base. In [18], they report flow visualization, aerodynamic force history and vortex breakdown location for a 75°-swept delta wing pitching between 25° and 50° in a ramp and hold motion. These are the results of Navier-Stokes simulations done at both laminar and fully turbulent conditions (Reynolds number based on chord, \( Re = 9200 \) and \( Re = 2 \times 10^6 \)). They have analyzed the effects of the pitch rate, pitch axis location and compressibility (Mach number \( M_\infty \) between 0.2 and 0.6).

Kandil et al. [19, 20] produced also remarkable results. Both Euler and turbulent calculations are presented for pitching delta wings in low subsonic flow (Mach number of 0.3) as well as transonic conditions (Mach number of 0.85). The sinusoidal oscillations have an amplitude of 2° in [19], while in [20] the coupled motion of pitching and rolling wing with an amplitude of 4° is simulated. Figure 8 shows the hysteresis loops obtained for a pure pitching wing.

![Figure 8: Lift and rolling moment coefficients for a pitching delta wing](image)

\( \alpha = 20^\circ + 4^\circ \sin(\pi t) \) [20]

Ekaterinaris and Shiff [21] report surface pressure distributions from a Navier-Stokes simulation of a double delta wing oscillating around 22.4° with an amplitude of 6.8°. This is for a low Mach number of 0.22 and a Reynolds number of \( 4 \times 10^6 \).

In Europe, Arthur et al. [22] compared results of Euler calculations (inviscid flow hypothesis) performed with codes from several European partners: DERA (now QinetiQ, U.K.), NLR (The Netherlands), Alenia Aeronautica (Italy) and DASA (Germany). Histories of aerodynamic coefficients and surface pressure distributions are examined for a 65°-cropped delta wing pitching in a sinusoidal motion of 6°-amplitude around two mean angles of attack: 9° and 21°. Fritz [23]...
has also studied this wing for oscillations around 9° and 15° with amplitude of 3° and 6°. Results with the Baldwin-Lomax turbulence model, proved to be more accurate than the Euler results.

These references addressing the case of numerical simulations of pitching delta wings are few compared to the amount of experimental reports. One important remark concerns simulations of wing pitching in sinusoidal oscillations: this is not the most studied case compared to ramp pitching simulations and when computed, the oscillations are small (4° in [20], 6° in [22]) in comparison to the experimental values (27.5° in [5, 12] for example). The present study, on the contrary, is focusing on larger amplitude oscillations similar to experimental conditions in order to study the whole dynamic stall phenomenon. The pitching experiment by Brandon [13] has been chosen as a test case because the amplitude of the oscillations presented there, is most suitable for a first numerical study, i.e. the range of angles of attack is not too high compared to all the other experimental references.

3.2. Mesh refinement and vortex flows

A lot of research has been made on adaptive mesh refinement. The main domain of application has been transonic or supersonic flow where shock waves need to be captured in the mesh to improve the accuracy of the results. For this purpose, sensors based on gradients of flow variables like pressure or density are usually very efficient. Such sensors also refine regions of stagnation like the leading-edge of a wing. However, these classical sensors are not well suited to capture vortices that also require a fine mesh. Surprisingly, only few studies are devoted to the refinement of flows containing vortices. These studies are mainly focusing on two research areas: delta wing flows and the flow around helicopter rotor blades. Some of them are presented below.

Pirzadeh [24] focuses on delta wing flows and shows results for 65° delta wings with different leading-edges and a wing-body configuration with chines. The refinement sensor used is the production of entropy and the refinement procedure is done by remeshing. In this paper [24], only the tetrahedra are refined (see Figure 9) although Navier-Stokes simulations on hybrid grids (with prisms in the boundary layer) are performed. Pirzadeh notes a large influence of the refinement on the vortex breakdown position and the surface pressure distribution below the vortex shows the typical increase in suction peak reached by refining the grid.

Entropy is also used by Modiano and Murman [25] to refine the mesh around a delta wing. The normal force coefficient predicted by Euler simulations is in better agreement with experimental data when adaptation is used. The position of the vortex breakdown also matches the experimental results.

Kang and Kwon [26] combine adaptations around shocks and vortices to im-
prove the simulation of a hovering helicopter rotor. The vortex is identified by a local maximum of vorticity and cells in the neighborhood of the vortex are divided. After six refinement steps, the vortex is captured up to 410 degrees of vortex age and the results show good agreement with the experimental data.

Helicopter rotor in a hover is also simulated by Duque et al. [27] with the help of two mesh-improving techniques: automatic grid adaptation to capture the vortex in the wake and overset structured grids for the rotor surface and tip flows. The sensor used to detect the vortex in this case is vorticity scaled with the local cell size to avoid grid clustering. The refinement is also limited to three subdivision levels. Here too, the vortex is traced in the wake up to two blade passages.

In [28, 29], Haines et al. present a method based on the eigenvalue analysis of the tensor of the velocity gradients to visualize vortices in a 3D flow field. Points where the tensor has one real and a pair of complex conjugate eigenvalues are in a vortical flow. The eigenvector associated with the real eigenvalue points in the direction of the vortex whereas the two other eigenvectors define the plane of rotation. Using this eigenvalue analysis, a reduced velocity representing the velocity in the plane of rotation can be defined and it is equal to zero at the core of the vortex. Later, this technique has been turned into a vortex sensor for mesh refinement. However, its use is not widespread and the three following references are the only examples known of the author (Paper D gives more details about the method).

In reference [30], a combination of an error estimate and a vortex sensor are used to refine the flow in the wake of a rotor blade. The solution is obtained using the finite element method to solve the Euler equations. The vortex sensor
is based on the eigenvalue analysis of the velocity-gradient tensor as presented above. Three test cases are presented and the refinement improves the resolution of the tip vortex which leads to better predictions of the thrust coefficient all cases.

This last vortex sensor is also used by Murayama et al. [31] to refine the grid around a 76° delta wing. Laminar computations at an angle of attack of 32° predict vortex breakdown over the wing only after two refinement steps. The sensor based on the eigenvalue analysis of the velocity-gradient tensor is compared to the second derivative of the total pressure that is also used as a sensor. The first one is deemed more efficient because of the limited number of cells it creates for the same level of accuracy.

Finally in [32], the sensor based on the eigenvalue analysis of the velocity-gradient tensor is applied to the simulation of high-lift wing configurations. The sensor is mainly active in the prismatic layer and capture the vorticity of the boundary layer. The tip vortex is not captured by the sensor.

In the present study, this last sensor is implemented and tested on several examples of vortical flows. The implementation follows the improved version of the method presented in [29] i.e. it is node-based and not cell-based. It is difficult to compare the implementations of the method in the previous three references because not so many details are given but the present study is unique in the fact that it highlights the role of the threshold value of the reduced velocity used to start the refinement. This threshold makes it possible to refine either a region concentrated around the core of the vortex (low values) or on the contrary a wider region around the vortex (large values). No other reference presents an analysis of the choice of the threshold.

4. Numerical methods

In this part, the two flow solvers used to carry out the present work are briefly described. As already mentioned, one, NSMB, is an "old" flow solver with a lot of features implemented, while EDGE, the unstructured code is less mature but computes on, in theory, more rapidly-generated and flexible grids. But before the two CFD programs are presented a few words are said about mesh generation.

4.1. Mesh generation

Mesh generation has been an important part of the work carried out for this thesis, especially in the last part. Indeed, before starting the computations, the first task of a CFD engineer is to generate a computational mesh around the geometry to study. And quite often this is a lengthy, experience-driven process that requires much more time than the computations themselves. This is particularly true for structured grids, the unstructured approach being more
automatic.

All the meshes used for the computations presented in this thesis have been generated with IC
cEM CFD\textsuperscript{3}. (The structured grid used in papers A and B has not been created by the author). IC
cEM Hexa has been used for the generation of structured grids. Its use requires a long learning process and a very good aptitude for “seeing” in 3D and design a feasible topology. On the contrary, IC
cEM Tetra used to create unstructured tetrahedra meshes is more user-friendly and rather easy to use (on “nice” geometries). All the initial unstructured meshes that have been refined by the mesh refinement methods developed in this thesis have been generated with Tetra and often use has been made of the so-called “density regions” that enable to locally refine the mesh.

A third mesh generator has to be named, the Tritet program [33] developed at the Swedish Defence Research Agency, FOI. This program has been tested for mesh generation around delta wings but most of its use comes from its adaptive mesh refinement capability that has been introduced as an adaptation module in the EDGE flow solver package (see section 4.3.2).

4.2. The structured flow solver NSMB

All the computations performed on pitching delta wings and reported in paper A and B, were obtained with NSMB, a Navier-Stokes Multi Block flow solver [34]. NSMB has been developed in a consortium composed of universities (EPFL, Switzerland; SERAM and IMFT, France; KTH, Sweden), one research establishment (CERFACS, France) and industrial partners (EADS-France; CFS-Engineering, Switzerland; SAAB Aerospace, Sweden).

4.2.1. General description

NSMB solves the Navier-Stokes equations on structured grids. The grids can be divided in several blocks (multi block). Patched grids are possible i.e., the interface between two blocks does not have to have a one to one correspondence, which makes it possible to have refined blocks in the computational domain. NSMB is parallelized using MPI as message passing language. In parallel computations, the load balancing is done at the block level.

The Navier-Stokes equations are discretized using the finite volume method. The equations are discretized in space with a 2nd or 4th-order central scheme with artificial dissipation or using one of several upwind schemes. For steady state computations, the time integration is achieved through the explicit Runge-Kutta scheme or the implicit LU-SGS (Lower-Upper-Symmetric Gauss Seidel) method. Several convergence acceleration techniques are available: local time stepping, implicit residual smoothing, multigrid (full multigrid available) and preconditioning for low-speed flows. Several physical models are available to

\textsuperscript{3}At the exception of the grid around the UCAV configuration
simulate the flow: inviscid flow (solving the Euler equations), laminar (viscous but non-turbulent), turbulent, LES (Large-Eddy Simulations), DES (Detached Eddy Simulations). Turbulence models include the Baldwin-Lomax algebraic model, the one-equation model of Spalart and Allmaras as well as two-equation models like Chien k-ε, Wilcox k-ω and Menter k-ω. Several level of chemistry modeling for air and nitrogen are available for hypersonic applications.

4.2.2. Moving grid approach

To simulate the pitching delta wing, the moving grid approach implemented in NSMB has been used. At the start of this study, the implementation had just been released and only tested in 2D cases for small amplitude oscillations on a deforming mesh (aerelastic applications). This study is the first test of moving grids in 3D and to the author’s knowledge the only application of NSMB to large amplitude motion.

The moving grid method is also called ALE (for Arbitrary Lagrangian Eule-
rian) approach because it considers a lagrangian grid (the motion of each point of the grid is known) in an eulerian fluid (the fluid is studied as a volume where flow particles move). Three levels of ALE are implemented: steady ALE on non-moving grids like when computing on a rotating frame, moving grids that are rigidly fixed to a moving body and deforming grids that allows small differential motion between the inner boundary on the body and the outer boundary at the far field. For this study, due to the large amplitude of the motion of the wing, only the method with a rigid grid is possible. The approach is very similar to wind tunnel testing: the mesh, fixed to the wing, moves in a constant free stream like a wind tunnel model pitches in a constant flow. The solid body rotation of the grid necessitates to recalculate the far field boundaries at each time step and the mesh velocity has to be taken into account in the flow equations and at the wing surface. Another possible approach is to change the free stream while the mesh and the wing are fixed but it appears to be less accurate for large amplitude motions. More details of the implementation of ALE in NSMB are included in references [35, 36].

In practice, computing the flow around a pitching delta wing is done in three steps:

1. The mesh is rotated to the mean angle of attack $\alpha_m$ (grids are usually made at zero angle of attack),

2. A steady-state calculation is performed to get a static solution of the flow at this angle of attack,

3. The unsteady-moving-grid computation is started with the static solution as initial conditions.
4.2.3. Dual time stepping

The dual time stepping method is used to integrate in time the time-dependent governing equations. The principle of the method is to have two time stepping loops: a real-time loop (also called "outer" loop) and a fictitious time loop (the "inner" loop). The solution at a real time step is obtained as a "steady state" solution of the inner loop: for each time step, a certain number of sub-iterations solves the equations for a fictitious time in order to get the solution for that time step. The advantage of the method compared to explicit time integration for example, is that there is no time step limitation (i.e. much larger real time steps can be used without any stability problem). In addition, the inner loop can be solved implicitly and all the acceleration techniques are available in the inner loop (local time stepping, multi-gridding).

This method is very useful for the present case with large amplitude motions that would require very small time steps with an explicit method. However, the method introduces several parameters very difficult to estimate a priori: the level of convergence in the inner loop (in relation with the number of inner iterations), the magnitude of the outer time step. There is no exact method to estimate these two parameters and the values are case dependent. Suitable values can only be found with a heuristic approach by a process of trial and error. The sensitivity of the results to these two parameters has been studied in this work.

4.3. The unstructured flow solver EDGE

All the work done on mesh refinement for vortical flows has been carried out on unstructured grids with the solutions obtained with the flow solver EDGE [37], developed at the Swedish Defence Research Agency, FOI.

4.3.1. General description

EDGE solves the Reynolds Average Navier-Stokes compressible flow equations (and Euler equations) on unstructured grids. The solver has an edge-based formulation and uses a node-centered finite volume technique to solve the governing equations. The edge-based formulation makes it possible to compute on any type of mesh: structured, unstructured (with tetrahedra, hexahedra, prisms or pyramids) or hybrid. In the code, only node and edge information is used. This information is provided by a preprocessor program that transforms the initial element information present in the grid file. The control volume required by the finite volume technique is obtained via a dual grid, the cells of which surround the initial mesh nodes (for example node $v_3$ in Figure 10). The dual grid is also created by the preprocessor program.

The spatial discretization is either central with artificial dissipation or upwind; both approaches are second order accurate. For steady state computations, the time integration is explicit with the Runge-Kutta scheme. Several convergence
acceleration techniques are available like agglomeration multigrid (with full multigrid capability to create a good initial solution), implicit residual smoothing and low speed preconditioning. Time-accurate calculations are computed either explicitly with a four-stage Runge-Kutta scheme or implicitly with dual time stepping (an explicit Runge-Kutta method is used within each real time step). In the version used for this work (version 3.1), two turbulence models are available, the standard Wilcox $k-\omega$ model and the Wallin & Johansson Explicit Algebraic Reynolds Stress model. More will come in the next version. The program has been parallelized using the MPI message passing library and can thus be run on computers with distributed memory like Linux clusters.

4.3.2. Automatic mesh refinement

A mesh adaptation module is included in the EDGE distribution. This module is a slightly modified version of a part of the mesh generator program Tritet [33] developed at FOI. In Tritet, the advancing front technique is used to create tetrahedra volume meshes with layers of prisms in the boundary layers. Two mesh refinement methods are available in Tritet: the remeshing technique that uses the full capability of the mesh generator to remove regions with too large cells and remesh them; the h-refinement technique that consists in dividing existing mesh cells. This latter technique is the one included in the EDGE distribution because it does not depend on a complete mesh generator. The meshing by subdivision is indeed relatively easy and does not require an advanced program. In addition, the meshing part does not require user intervention (no graphical user interface required) and the geometry is obtained from the initial mesh (no handling of the CAD geometry since the new points on the boundaries are not projected on the exact geometry).

The refinement is edge-based and depending on the number of edges marked for refinement, the tetrahedra cells are either divided in two, four or eight new cells. These restrictions are imposed to avoid creating ill-deformed cells. Only one level of subdivision is allowed per refinement. A minimum cell size is specified to avoid refining ad infinitum. For viscous hybrid grids with layers of prisms near the wall surfaces, only the tetrahedra elements are taken into account for refinement. If necessary, the cell division is prolonged through the prism layers by dividing those until the wall is reached. To improve the quality
of the cells when performing several refinement steps, potential stretched cells resulting from the refinement are further refined in the longer direction.

At the beginning of this work, only one sensor was available to detect regions that require refinement. This sensor is based on the gradients of the density, the three components of velocity and the pressure. If the gradients between two connected nodes are larger than a threshold value, the edge between them is marked for refinement, which leads to the cell division. This sensor gives excellent results to refine meshes around shocks or leading- and trailing-edge regions for airfoil or high aspect ratio wing simulations (see Figure 11).

![Figure 11: Refined mesh around the RAE2822 airfoil at M=0.734](image)

4.3.3. Implementation of sensors for vortex flows

A part of the work presented in this thesis has consisted in developing and implementing mesh refinement sensors that capture vortices in a flow solution. Indeed, the initial sensor presented above is rather inefficient for vortex-dominated flows: the refined cells are clustered around the walls and wing edges where the gradients are high and only a small number is found in the separated vortical regions. In order to refine the regions around vortices, three new sensors have been developed, implemented and tested in the EDGE adaptation module.

These sensors are based on visualization techniques used to highlight vortices in a CFD solution. The first one is based on the production of entropy, often used to represent vortices as surfaces of isovalue (so-called isosurfaces). As a refinement sensor, it helps to mark all the edges of the mesh that are located inside an isosurface with a threshold value. The second sensor is the ratio of the total pressure in the flow to the total pressure at the far field and operates in the same way as the previous sensor. The third sensor is based on the analysis
of the velocity field to detect vortices. More exactly, it requires an eigenvalue analysis of the tensor of the velocity gradients at each node (also called rate-of-deformation tensor). The tensor is constructed at each node in the mesh from a postprocessing file containing the gradients of the primitive variables. Nodes for which the tensor has one real and two complex conjugate eigenvalues are in a vortical flow. Those that in addition, have a low reduced velocity are close to the core of the vortex. This method makes it possible to identify the nodes in the vortex core and then mark the connecting edges in order to divide the surrounding cells. All these developments and implementations have been tested on a number of vortex-flow cases and results are reported in the appended papers, in particular paper D gives more details about the method.

5. Summary of appended papers

5.1. Paper A

Results of the simulations of a 70° delta wing pitching to high angles of attack are presented in this first paper. The angle of attack is changing in sinusoidal oscillations with large amplitudes, the main test case being oscillations around 22° with a semi-amplitude of 18°. Several other cases are also included (see Figure 12). The results are shown in the form of plots of aerodynamic coefficients versus the angle of attack like Figure 12. The hysteresis loops observed in experimental conditions are well reproduced by the simulations. Studies of the periodicity, the choice of the timestep and of the parameters for the dual timestepping methods are reported. Most of the computations are inviscid and laminar results show little difference except at low angles of attack where the boundary layer is believed to play a more important role (see Figure 13).

![Figure 12: Normal force coefficient for the pitching delta wing](image1)

![Figure 13: Comparison of laminar and inviscid results](image2)
5.2. Paper B

This study is the continuation of the first one. It deals with simulations of a pitching 70° delta wing too. Visualizations of the formation of the vortex and its decay as the angle of attack changes are reported. The delay in the motion of the vortex breakdown is illustrated. Figure 14 for example shows the vortex as an isosurface of total pressure at a constant angle of attack of 22° but for different phases of the oscillation. A few results of turbulent calculations are added to the inviscid ones.

![Visualization of the vortex at \( \alpha = 22° \)](image)

Figure 14: Visualization of the vortex at \( \alpha = 22° \)

In a second part of the paper, a method to process the data resulting from the calculations while still computing is presented. This co-processing method
(as opposed to post-processing) is aimed at reducing the amount of data saved to disk for such a large test case as the simulation of a pitching wing. The coupling between the flow solver NSMB and the visualization software EnSight is explained.

5.3. Paper C

In this study, the influence of the refinement of the grid in the vortical region of the wake of a delta wing is investigated. Several unstructured meshes are created with refined regions, above and downstream of a semi-span 70°-delta wing. The length of the refined region is varied from stopping at the trailing-edge to extending one root chord downstream of the trailing-edge or extending all the way to the far field boundary. Figure 15 shows for example the mesh with a refined region that is one root chord long (the wing lies in the middle of the figure).

![Symmetry plane of the mesh with a short wake refinement](image)

Figure 15: Symmetry plane of the mesh with a short wake refinement

The refinement is shown to have an influence on the location of the vortex breakdown which in turn affects the surface pressure distribution of the wing. It is concluded that the wake has to be refined in order to get accurate results. However, a short refinement is sufficient and more cost effective than the full refinement that extends to the far field boundary.

The second part of the paper focuses on wind tunnel corrections and try to estimate the influence of the wind tunnel walls on the vortical flow. A grid for the 70°-delta wing inside the wind tunnel where it has been tested [38] has been
created (see Figure 16). A short review of wind tunnel corrections applied to delta wing flows is given. The analytical method by Traub [39] is emphasized because it takes vortex breakdown into account. A corrected angle of attack is calculated with this technique for the 70° delta wing in the wind tunnel. The results obtained with the wing in the wind tunnel are compared to the ones for the wing in corrected free air conditions. The improvements seen in the front part of the wing are ruined by a prediction of the breakdown too far upstream. The conclusion is that a modified angle of attack is not enough to correct for the wind tunnel walls when vortex breakdown is present.

![Figure 16: View of the mesh with the wing in the wind tunnel](image)

5.4. Paper D

In this paper, three sensors for identifying vortices in a flow field and then automatically refine the computational grid are presented. The two first sensors are similar in their implementation and both based on easily-calculated flow quantities: the production of entropy and the ratio of the total pressure to the total pressure at the far field. Nodes with a lower (total pressure) or higher (entropy) quantity than a defined threshold value are marked for refinement. The third sensor identifies vortices through an eigenvalue analysis of the tensor of the velocity gradients. Nodes with one real and two complex conjugate eigenvalues as well as a low reduced velocity are close to the vortex core and thus marked for refinement. The refinement method itself is by subdivision of the existing cells, a so-called h-refinement. Here, all the edges linking two nodes marked by the sensors, are divided. The sensor with the eigenvalue analysis is first tested on artificially-created vortices in order to understand and calibrate its functionality.

The second part of the paper is about simulations of delta wing flow with automatic mesh refinement performed using the three sensors presented in the first part. The three sensors give satisfactory results with this test case but they
have slightly different properties: the first two refine a large region around the vortex above the wing and in the near wake, whereas the third sensor focuses on the vortex core and captures nodes far downstream of the wing (see Figures 17 and 18).

![Image]

Figure 17: Nodes marked for refinement by the sensor based on the eigenvalue analysis of the gradient of velocity tensor.

The three sensors are also compared through a study of the sensitivity to the chosen threshold value. The two sensors based on the flow variables (entropy and total pressure) are very sensitive around their respective critical value (0 and 1) and able to mark the whole mesh for refinement. On the contrary, the third sensor has a smoother response to the change of threshold and is limited to the vortex.

5.5. Paper E

A study of the wake of a 70° delta wing is presented in this paper. Euler simulations are performed on a mesh that is initially refined in the wake of the wing. The results show that a pair of vortices interact in the wake of the half-span wing: the main leading-edge vortex and a vortex that is formed by the roll-up of the vortex sheet shed from the trailing-edge. This interaction is rarely seen in the simulations of delta wing flows because the standard meshes are too coarse in the wake region. Mesh refinement with the sensor based on the eigen-
value analysis of the tensor of the velocity gradients is used to further improve the grid resolution along the path of the vortices. The sensor captures very well both vortices until they reach the boundary of the domain. Results with the finer grid resolution agree reasonably well with experimental data. The comparison is based on contours of total pressure coefficient in crossflow planes in the wake of the wing. Streamline visualization shows vortices spiraling even far downstream of the wing (see Figure 19).

Simulations and one step of refinement are also performed on a full-span wing to confirm the position of the second vortex above the leading-edge vortex. Small differences are shown. Finally, a study of the influence of the artificial viscosity on the convection of the vortices is carried out. It highlights the sensitivity of the results to the fourth order dissipation.

5.6. Paper F

In this last paper, use is made of the experience accumulated during the previous studies and the tools developed to simulate the aerodynamics of a UCAV. The configuration is the Saab Aerosystems U512A-NUK14 configuration. Steady state Euler simulations are performed for a range of angles of attack from 0° to 27°, with and without sideslip (β = 15°). The results, in the form of isosurface of the total pressure coefficient, reveal that the aerodynamics of the UCAV is dominated by the formation of a pair of vortices on the front
Figure 19: Streamline visualization of the system of vortices on a refined grid part of the fuselage. The footprints of the vortices are visible in the surface pressure distribution and it is seen that the pressure on the lifting surfaces is also influenced by the vortices. Grid refinement with the vortex sensor based on the eigenvalue analysis of the tensor of the velocity gradients is performed to improve the grid resolution in the vortical region. The pressure distribution obtained after refinement is slightly changed.

Computations with an angle of sideslip of 15° show an unexpected interaction of the body vortices with the V-tail (see Figure 20). Depending on the angle of attack, the vortex passes below or above the tail or directly hits the tail. The presence of the vortices and the interaction with the control surfaces result in non-linear aerodynamic characteristics that the simulations reproduce rather well up to a critical angle of attack. Above that angle, the predicted stability coefficients are very sensitive and deviate from the experimental data. Mesh refinement slightly increases this critical angle. For higher angles of attack, the convergence histories show that the flow is unsteady, partly explaining the scattered results.

6. Discussion

In this part, comments about the work presented in the appended papers are made and the papers put in relation with each other.

The simulations of the pitching delta wings in paper A and B succeed in capturing the hysteresis loops of the aerodynamic coefficients and so far has not been matched in terms of amplitude of oscillation by more recent publications to the knowledge of the author. Even if simulations of full aircraft in maneuver
Figure 20: Vortex visualization around the UCAV configuration at $\alpha = 15^\circ$ and $\beta = 15^\circ$.

have been reported since (see for example [40]), the maneuvers are still limited to a maximum semi-amplitude of $10^\circ$ and the results do not match the experimental results very well. So, as initial results, these simulations were successful and ground-breaking at the time of their publication. They were an initial demonstration that simulation of dynamic lift with large amplitude motion is possible, thus expanding the envelope of the methods, limited to small amplitude motion until then. In addition, the computations reached the limit of the computer power available then (see following remarks on computer power). With today's methods and computer power these results could be improved. Mesh refinement could improve the grid resolution in the vortical flow region. Later improvements to NSMB have made it possible to have refined patched grids which could be placed above the wing and in the wake. Computing on a finer mesh would hopefully decrease the observed grid sensitivity of the results on the rather coarse mesh of paper A and B. Although, in that case a coarse mesh might have helped reaching convergence by filtering out the small scale flow features from the simulations, at the expense of the accuracy.

The choice of the test case and of the experimental data to compare with is subject to discussion. It has not been easy to choose an experimental test case that expanded the oscillation amplitude beyond what was computed before, but was not too large so that a solution could be attained. Beyond the fact that all the reported experimental work are not complete as explained in the literature review part, the well documented results are for very large pitching
oscillations. Experiments by Soltani and Bragg [5, 12] were considered because the history of several aerodynamic coefficients was reported but oscillations between 0° and 55° were deemed too large. The 18° semi-amplitude oscillations of Brandon appeared better although reference [13] does not contain so much information about the exact geometry of the wing and the set-up used in the experiment. Personal communication with the author of the experiment revealed that the wing had not exactly the same leading edge geometry as the computational model and the pitching axis was slightly below the one used for the computations. In addition, the author of the experiment did not consider the results as very reliable. In these conditions, an exact comparison with the experimental data is difficult and one should not expect the results to show more than trends and give more than a good insight of reality.

A more general remark, about the work carried out through the last five years, concerns the increase in computer power. Simulations in paper A and B were computed on one processor of a Fujitsu VX machine that has a peak performance of 2.2 Gflops per processor. In comparison, the simulations in the last paper were carried out on an Itanium 2 Linux cluster, with a peak performance of 3.6 Gflops per processor and up to 16 processors have been used in parallel. Of course, NSMB on the vector machine certainly reached higher performance than EDGE on a single processor of the Itanium 2 cluster but the computations still took something like five to ten times more time. Steady state computations on the 700,000 cell structured mesh took in the order of two hours, whereas it would take less than half an hour wall clock time on the newer computer. Simulations performed in the last paper could probably not have been carried out on the Fujitsu machine due to a lack of memory. With today's computer power, the work in paper A and B would be done much more rapidly and certainly more accurately on a finer mesh. These unsteady simulations of maneuvering wing which were at that time too expensive and complex for being useful to the industry, might now be more feasible and practical.

A few words have to be written about the change of flow solver and the duality structured/unstructured grids. As mentioned earlier, in paper A and B no mesh refinement or sensitivity study was presented, mainly because of the cumbersome process of mesh generation for structured grids. When the sensitivity of the results was discovered, hopes were placed on the unstructured approach instead. Indeed, the possibility to refine the mesh in the vortex and the rather “easy” generation of meshes were attractive. However, suddenly the large freedom in the choices to design an unstructured mesh raised more questions than it gave answers. With structured grids, much of the effort was put on adapting the mesh topology to the geometry, refining the boundary layer, avoiding singularities and making sure that the lines that extend to the far field boundaries did not penalize too much the always growing number of cells. With unstructured grids, many of these problems disappeared, and the effort had to be put on finding the locations where fine cells best improve the accuracy of the results (since in this case too, the number of cells is limited).
This question is what motivated papers C and D.

The results on the refinement of the wake in paper C were rather inconclusive, certainly because of the presence of vortex breakdown in the solution. An easier approach should have been to discard angles of attack where vortex breakdown appeared on the wing and start the study of the mesh at lower angles of attack instead. The results presented were also one of the first the author computed with EDGE and experience now tells us that the artificial dissipation should have been reduced and its effect analyzed more closely.

Paper E is, to the knowledge of the author, unique in the accuracy of the results computed. Lee and Lan [41] do report results of simulations of delta wing wakes but on a mesh much coarser than the one used in the present study. The results presented in paper E contribute to the understanding of the basic flow around a delta wing. Although limited to inviscid flow, the results give an interesting view on the dynamics of the wake with the interaction of the two vortices present there. The second, weaker vortex forming at the trailing-edge of the wing has so far received very little attention and is seldom captured in the published numerical simulations of delta wing flows. Indeed, the mesh that are commonly used are too coarse in the wake region and if the mesh is refined, only the stronger leading-edge vortex is captured because the trailing-edge vortex does not appear on the coarse mesh and is hence invisible to the sensor. The presence of the vortex forming at the trailing-edge certainly influences the leading-edge vortex, at least where the trailing-edge vortex rolls around it. It might be necessary to accurately capture this latter in the solution to get an exact simulation of the leading-edge vortex close to the trailing-edge of the wing (both in term of position of the vortex and exact flow characteristics). In particular, it is known that vortex breakdown is sensitive and an exact prediction of the on-set of breakdown over the trailing-edge of the wing might require the presence of the second vortex in the solution.

Paper F is a clear application of research methods to an industrial subject. The short time (less than three weeks) in which most of the results presented there were obtained, is also illustrative of the industrial approach. The fact that the mesh refinement methods worked on such a large mesh and improved the results of the aerodynamic coefficients beyond the critical angle of attack where unsteadiness starts is encouraging. The results, though preliminary, give a good insight into the vortex interactions that occur in sideslip conditions. The flow visualization pictures showing the body vortices hitting the V-tail are valuable results for the designer, especially since they are difficult to get from wind tunnel experiments. These interactions can help understand the complex non-linear aerodynamics of the UCAV. The unsteadiness that starts at a critical angle of attack prevents the current steady state Euler simulations from predicting accurately the aerodynamic coefficients at large angles of attack. However, the simulations give a warning signal that vortex shedding or maybe vortex/structure interactions occur and require additional attention. As
concluded in the paper, unsteady simulations should definitely be performed to improve the aerodynamic predictions at large angles of attack.

7. Concluding remarks

The work presented in this thesis has hopefully clarified some aspects of the vortical flow over delta wings. It has shown that simulations of maneuvering vehicles are now feasible and give a good insight into the dynamic characteristics of the vehicle. The new tools for automatic mesh refinement were developed to satisfy the industrial needs of accurate simulations performed in a rapid and efficient way. Current UAV designers for example, can now use these tools with confidence and be sure of the improved accuracy of the results. In addition, the research work on unsteady simulations has ushered in this technique that should start to appeal to the industry when trying to solve a complex time-dependent problem that is known to be inaccurately simulated with standard steady-state methods. These methods will certainly continue to improve and make CFD simulations both more accurate and more efficient.

8. Future work

In this part, ideas for future work on a similar subject as the one treated here are presented. It could be the basis for a complementary work on the subject. The paragraph can also be seen as a “wish list” of features that ought to be added or improved in the flow solver EDGE in order to get more accurate/efficient simulations of vortical flows. (The flow solver NSMB is not mentioned here because most of the ideas are for unstructured grids and the others are already implemented in the program ...).

Mesh refinement Concerning mesh refinement, a first improvement could be to better integrate the h-refinement module with the flow solver EDGE. A first step would be to enable a fully automatic mesh refinement procedure for steady state simulations i.e. that refinements be done automatically every 500 iterations for example, using predefined parameters. This approach would be very useful for parameter studies or small changes in the geometry like the flap settings for simulations of a high-lift configuration. At present, several programs have to be used necessitating user input even if the refinement parameters are known already at the start of the computations. To achieve that, several programs would have to be linked to EDGE, like for example the preprocessor and the program that sets the boundary conditions.

While using the h-refinement module, the number of refinement steps has been limited because stretched cells are created and worsen the quality of the refined mesh. A typical case is when a cell is divided in two and one of the resulting cells is again divided in two at the next refinement step. A fix to the problem has been implemented but it only slightly improves the quality and defers the
problem to the next refinement step. A better solution is to keep track of the history of the refinement in the mesh file, so that it is possible to return to the initial undivided cells and refine them more than originally done.

Coarsening would also be a very useful tool when trying to improve the mesh quality at a reasonable cost. Still with the h-refinement technique, the preservation of the exact geometry when adding nodes should be considered. That requires that the CAD information follows in the mesh file and that the refinement program is able to work with it.

In relation to the work presented in this thesis, mesh refinement with the remeshing technique should be investigated when using the sensor based on the production of entropy or the one based on the ratio of the total pressures. Indeed, these two sensors mark regions that are closed volumes and are well suited for this technique. The advantage of the remeshing technique over the h-refinement is that smaller cells can be created already during the first refinement step, reducing the number of refinement steps required. The new mesh should also be of better quality with smoother transitions between cells.

Finally, unsteady automatic mesh refinement should be considered. This will certainly become a necessity in the future. H-refinement is usually preferred in this case because it is fast and robust.

Higher order spatial schemes All the results presented in this thesis have been sensitive to mesh resolution. Higher order spatial discretization should attenuate this sensitivity and either make it possible to compute on coarser grids for the same level of accuracy or increase the accuracy of the results on the current grids. Although it is not a requirement for vortex flow simulations as it is for aero-acoustics for example, this type of flow simulations would anyhow benefit a lot from the implementation of a higher order spatial discretization in EDGE.

Moving grids The moving grid approach could be implemented in EDGE to simulate maneuvering vehicles. The approach has made its way into “old” structured flow solvers and is mature enough to be adapted to an unstructured solver. More and more advanced coupling between CFD and flight mechanics are presented in the literature, which often requires the simulation of a maneuvering vehicle.

All the different techniques mentioned above, would of course lead to the simulation of unsteady pitching delta wings with mesh refinement on unstructured grids. With that, then, maybe more accurate results could be obtained without the huge computer power that a globally refined mesh requires.
Wind tunnel interference. Paper C presents results on the influence of wind tunnel walls on delta wing flow. A recent Ph.D. thesis [42] also treats the subject but leaves many questions unanswered. A lot more work can be carried out on this subject with CFD as a tool. Especially with unstructured grids, it is now rather easy to add the walls or the support system in a mesh and study their influence on the flow field. The presence of vortex breakdown in particular invalidates the classical correction methods and no accurate method for correction remains in that case. It is possible that the flow becomes so complicated that no general method can be devised. In that case, CFD simulations might be the only answer and guidelines or practices would have to be established.

CFD simulations of delta wing flows. This thesis has already presented a lot of results concerning delta-wing flow simulations but more could be done and ideas are not limited, only time and money are limiting factors. An accurate simulation of the separated flow over a pitching delta wing will require an advanced flow modelling. Detached Eddy Simulations (DES) is one of these advanced models that shows great potential at a cost that is becoming acceptable. DES simulations of a pitching delta wing would certainly contribute to the better understanding of the flow around a maneuvering wing. To start with, pitch and hold motion that is less demanding than the periodic pitching oscillations, could be simulated.

The interesting dynamics of the wake of a delta wing that is treated in paper E also requires further work. The results in paper E are only initial. Here too, a more advanced physical modelling should be tried. In addition, the influence of the wing geometry should be investigated, in particular the shape of the edges. Does the trailing vortex also form at a blunt trailing-edge? And what about the leading-edge, does a rounded edge influence the formation of this vortex? Reynolds number effects might also be worth studying, indeed the vortices might merge at lower Reynolds numbers for example.
References


