U-Kite development

A Thesis Submitted to the Department

of

Mechanical Engineering

at the

Galway-Mayo Institute of Technology

by

John Robert Doyle

In Partial Fulfillment of the Requirements for the Degree

of

Master of Engineering

April 2000



ABSTRACT

The Following thesis describes the work which was undertaken in the conception, development, and the construction of an open sea towed vehicle used to deploy a wide range of oceanographic monitoring equipment. This streamlined submersible vehicle, is towed alongside a ship or small vessel, and remains slightly buoyant at all times, but once it is moving it uses its wings which are configured upside-down with respect to those on an aircraft to pull it down to depth and move it away from the towing vessels wake and wave system, while assuring lateral stability. The Preliminary design requirement called for a light, robust, stable, simple, and low cost platform able to operate in a close to rough sea surface, and can be worked from a relatively small vessels such as those used for surveys of estuaries and coastal. During the design phase of this project, tools such as parametric feature based solid modeling, computational fluid dynamics, and finite element analysis were employed to provide a solution to the problem. A prototype was built using CADCAM technology, and strong composite materials.

ACKNOWLEDGEMENTS

I would like to recognise and thank the following individuals, Dr. Patrick Delassus for the time he spent helping me to make this work my best effort. I would also like to thank Dr. Marcel Cure for his invaluable direction, and the other post graduate students in the department, who in one form or another provided assistance, namely James Conerney, Richard Maxwell, and Carine Gachon. Finally, I'd like to thank the department's technical staff Gabriel Costello, Pio Kerrane, and Michael Keaveney for their endless assistance and cheerful source of motivation.

TABLE OF CONTENTS

CHAPTER ONEIntroduction1.1 Historical background1.2 U-Kite the concept1.3 Brief description of drawings1.4 Description of concept	2 2 4 5 6
CHAPTER TWO Fluid flow basic concepts	9
2.1 Introduction	10
2.2 External flow	. 10
2.3 Viscosity	11
2.4 Generation of lift	12
CHAPTER THREE Design methodology and tools	17
3.1 Introduction	18
3.2 Design methodology	18
3.3 Pro/Engineer	19
CHAPTER FOUR Redesigning the U-Kite	22
4.1 Design brief	23
4.2 Vehicle design	24
4.3 Design constraints	25
4.4 New design	26
4.5 Lifting line theory	29
CHAPTER FIVE Material selection	39
5.1 Introduction	40
5.2 Buoyancy	40
5.3 Aluminium and aluminium alloys	42
5.4 Fibre reinforced plastics	43
5.5 Design considerations	44
5.6 Design flexibility	44
5.7 Fibres	44
5.8 High strength glass fibres	45
5.9 Corrosion resistant glass fibres	45
5.10 Fibre cost	46
5.11 Forms of reinforcement	46

Page

5.12 Mats	47
5.13 Resins	49
5.14 Thermosets	49
5.15 Fillers	50
5.16 Laminate design	50
5.17 Hand lav-up	52
CHAPTER SIX Mould fabrication	55
6.1 Introduction	56
6.2 Computer numerical control	56
6.3 The principle of NC program generation	
based on a CAD model	58
6.4 Using Pro/Manufacture	60
6.5 Pro/NC-Check	62
6.5 Pro/NC-Post	.63
6.7 Tooling	64
6.8 Fabrication of mould tool for hydrofoil fairing	65
CHAPTER SEVEN Computational fluid dynamics	75
7.1 Introduction	76
7.2 Viscous incompressible fluid flow with inertia effects	77
7.3 Finite element method	81
7.4 Finite element modeling.	82
7 5 Turbulence modeling	83
7.6 Definitions and orders of magnitude	80
7 7 FLOTRAN	
7.8 Turbulence models	
7.9 Fluctuations and time-smoothed quantities	82
7.9 The requestion Snalding-Launder model	
7.10 Two equation Spating Database model model	86
7.17 Doundary Conditions. 7.12 Finite element method and equations	87
7 13 3D elements	
7 14 Shape functions	. 91
7.15 Discretization of equations	
7 16 FLOTRAN analysis	
7.17 Three-dimensional CFD analysis of the U-Kite	102
7.18 Geometry definition, and Model prepartion	103
7.19 Results	. 107
7.20 Design of upper hydrofoil	110
· · · · · · · · · · · · · · · ·	
CHAPTER EIGHT Conclusions	114
8.1 Field Trials	. 115
8.2 Conclusions	. 115

. . .

10

v

LIST OF REFER	ENCES	119
APPENDICES		
Appendix A	Hydrofoil definition	A1
Appendix B	U-Kite Drawings	B1
Appendix C	Assorted photographs	C1
Appendix D	Lifting Line theory Mathematica Notebook	DI

vi

Chapter 1

Introduction

1

1.1 Historical background

Oceanography is the science of the oceans, their interaction with the atmosphere above and with the underlying sea-floor sediments and oceanic crust, their chemical and biological components, their physical properties and geology. Oceanography relies heavily on a range of technologies, electronics, optics, and acoustics which together with the engineering involved in the design and construction of instrument platforms, winches and wires enable oceanographers to make observations and sample the remote ocean depths. To make such observations oceanographers require elaborate and robust equipment to survive the hostile environment. The deployment and the recovery of instruments from ships at sea, depends heavily on seamanship, and the efficient use of winches and hoists. There are risks involved, particularly in rough sea conditions when working on deck is difficult if not impossible, and delicate instruments may be damaged in handling, and as they are lowered over the side, perhaps swinging before entering the water.

Once in the water, hazards to instruments are present. The design must be appropriate to the conditions in which they operate which may be at high pressures, salty and corrosive. Near the surface instruments are subjected to large and fluctuating forces applied by breaking waves. Traditionally oceanographic measurements were taken from stationary ships, which was very costly in ship time.



Figure 1.1 Aquashuttle



Figure 1.2 Nv-shuttle

To save time, engineers and scientist have designed and built instruments and vehicles that may be towed behind or alongside a moving ship to allow continuous real-time data collection. Data obtained from are transmitted back to the towing vessel via RS232 and RS422 communications. Examples of such vehicles include the Aquashuttle and the Nv-shuttle (Chelsea Instruments Ltd).

The towed platform described in this thesis was designed as a vehicle to carry fast responding sensors at depths up to 40 metres from a vessel traveling at speeds of up to 10 knots per hour. This system can produce continuos records, of the parameters sensed, and can be worked from relatively small vessels such as those used surveys of coastal waters and estuaries.

The design of a body to move through any fluid is a complex problem. Although the theory is well established for fixed wing aircraft there has being relatively little research into the design of bodies towed through water. While there are similarities between the aerodynamic, and hydrodynamic cases, the use of a towing cable from a moving support vessel, are very significant and increase stability problems.

Ideally such vehicles should be self-propelled (i.e. independent of a surface vehicle), but the cost of a small power plant and an underwater guidance system for such vehicles would be prohibitive. Towed systems are widely used and will be continued to be used in near future until a cost effect alternative is found.

In 1995, Seasense Ltd. together with the marine institute at the National University of Ireland, Galway embarked on the development of a new oceanographic towed vehicle. The aims of the project was to:

(1) Provide an apparatus for controlling the lateral and vertical displacement of oceanographic instrumentation.

(2) Provide a towed underwater apparatus which is highly stable over a wide range of speeds and which is capable of assuming a predetermined orientation regardless of its initial position.

(3) It is a further object that such device displaces the instrumentation while creating a minimum of turbulence and noise.

(4) Another object is that such device be durable and small enough to be deployed and retrieved with the cable, without requiring attachment or removal during either deployment or retrieval.

Prototypes were designed and subsequently built. These proved successful during trials in Galway bay, but would need to be re-engineered if the vehicle was every to become a saleable product.

1.2 U-Kite the concept

The U-Kite has characteristics in common with other of devices, most notably paravanes; Paravanes displace towed cables in a lateral direction away from the path of a towing craft. Paravanes are towed bodies affixed at the end or along the length of a towed cable to position away from the path of the towing craft. A fin or vane on the paravane causes a lateral displacement of the cable by producing lift in the lateral direction.



Figure 1.3 Bi-winged paravane.

Paravane systems depend on components other than the vane itself to set and stabilize the direction of the hydrodynamic force. The stabilizing components used on paravanes include bridles surface floats on tethers, net floats, and trawl chains. These components increase size; weight, drag, and noise thereby limiting use to low speed operations. Because many paravanes are large or mechanically complicated devices they must be removed from the water separately as the tow cable is retracted. Larger paravanes require the use of a hoist to remove them from the water. More complicated paravanes are generally less sturdy and must be handled with care to avoid damaging them and their inner workings.

1.3 Brief description of drawings

A complete understanding of the U-Kite, and many of the accompanying advantages will be readily appreciated as the concept becomes better understood by reference to the following detailed description, when considered in conjunction with the accompanying drawings (N.B. drawings are of the initial prototype as conceived by Dr Marcel Cure).



Figure 1.4 shows a vessel towing a cable with a U-Kite

5

1.4 Description of concept

In the following text, all references to lateral mean in the direction orthogonal to the direction of tow. Referring now to figure 1.4, and figure 1.5, The U-Kite, which resembles an airplane. The towed underwater apparatus consists of a cylindrical fuselage or body; wing members secured to the fuselage. By using two wing members, disposed at suitable angles to the vertical axis of the U-Kite, a dihedral effect is produced which is highly efficient in creating the hydrodynamic lift necessary to maintain the vehicle at the desired depth and lateral offset from the towing vessel. Lateral and longitudinal stabilizers are attached to the tail portion of the fuselage, for maintaining the fuselage at a proper operating orientation in the water and for maintaining the U-Kite correctly lined up with the flow. The apparatus also consists of a bridle¹.



Figure 1.5 shows an isometric view of a U-Kite.

¹ The bridle of the U-Kite is the wire/line to which the tether is attached. The bridle is attached to the U-Kite at two locations to spread the lift force generated (see Fig. 1.4).

As generally shown in figures 1.4 through to 1.6, the fuselage, comprises an elongated tubular casing having a hemispherical nose portion for maximizing the internal volume of the fuselage and for providing a streamlined profile in the water. Such internal void spaces not only serve to accommodate instrumentation, but the void spaces may be ballasted with various lightweight materials, such as foam, such that the U-Kite will be neutrally buoyant in water.

Disposed upon the extremity of each wing, away from fuselage are ballast pods. The ballast pods extend for a slight distance perpendicular to the wing surface thereby channeling fluid flow over the wing surface to increase the effective aspect ratio of wings and prevent vortex shedding off the wings. The ballast pods are disposed at the extremity of wings to provide a buoyant force differential to maintain the orientation of the U-Kite. The ballast pod mounted on upper wing has positive buoyancy in water. The ballast pod mounted on lower wing has negative buoyancy. Thus the differential in buoyancy between upper ballast pod and lower ballast pod provides a righting moment to maintain the U-Kite in the near vertical position and the use of stabilizers prevents the fuselage and wings from spinning.

In operation, cable is towed from a vessel causing water to flow by device. The hydrodynamic flow over angled wings creates a pressure differential between one side of wings and the other side resulting in a force being exerted on cable.



Figure 1.6 A front view, of the U-Kite.

The orientation of device maintained by the difference in buoyancy between upper and lower ballast pods, and by the dihedral force generated by the wings.

Referring now to figure 1.7, there is shown a U-Kite towed on a cable. When the vessel tows the U-Kite through the water, the wings of the device produces hydrodynamic lift port or starboard of the towing vessel.



Figure 1.7 Shows, a two dimensional free-body diagram of the forces acting in the lateral direction.

Chapter 2

Fluid flow, basic concepts

2.1 Introduction

Hydrodynamics is the study of forces acting on an object. These forces become active when an object moves through the water. It is important to understand these forces in order to design a vehicle such as the U-Kite. In this chapter the underlying principles of fluid dynamics are reviewed, much of the material presented in here can be found in introductory texts on subjects such as fluid mechanics, fluid dynamics, and ship design. Much of this material can be found in [1,12,15,16,17].

2.2 External flow

The motion of water around any body produces velocity, and pressure variations, which produce hydrodynamic forces, and moments. It is customary to resolve the resultant force on the body into a component perpendicular to the free-stream velocity direction (called lift), a component parallel to the free-stream velocity direction (called drag), and the third component, which is perpendicular to both lift and drag (called side force). The moments about all three coordinate axes (as shown in figure 2.1) are designated as yawing, rolling, and pitching moments.



Figure 2.1 Hydrodynamic forces, and moments.

The magnitude of the forces and the moments that act on the body depend on the combined effects of many different variables. The parameters, which govern the magnitude of hydrodynamic, include:

- 1. Configuration geometry.
- 2.Orientation.
- 3. Body size.
- 4. Free-stream velocity.
- 5. Water density.
- 6. Reynolds Number (as it relates to viscous effects).

The calculation of the hydrodynamic forces and moments requires that the engineer to be able to relate data obtained from other flow conditions to the condition of interest. Thus, data from flow tank or wind tunnel tests are often used, where scaled models are exposed to flow conditions that simulate the design environment. Alternative methods of determining hydrodynamic forces and moments involve using mathematical techniques to model the flow field. Such techniques include the finite element method, the finite difference method, boundary element method, and lifting line theory, conformal mapping etc.

2.2 Viscosity.

All fluids have viscosity that causes friction. Viscosity is a measure of the fluid's resistance to shear when the fluid is motion. Viscosity is the proportionality between shear stress and the velocity gradient. The ratio of viscosity to mass density is referred to as the kinematic viscosity. The proportional relationship between the shear stress and velocity gradient is known as the Newtonian relationship.

A basic concept of fluid mechanics is the dimensionless Reynolds's number.

VL

oVL

Where

V = Velocity

L = Length

 ρ = Density of fluid

 μ = Viscosity of fluid

v = Kinematic viscosity

This parameter correlates the viscous behaviour of all Newtonian fluids. For example, if two different situations have the same Reynolds number the fluid flow will similar. Therefore, the behaviour of a flow field around a deeply submerged hydrofoil is equivalent to that around a geometrically similar isolated airfoil operating under dynamically similar conditions.

2.3 The generation of lift.

The lift force generated by a hydrofoil (or airfoil) is defined as the component of the hydrodynamic force, which is perpendicular to the direction of the oncoming mainstream velocity vector. It may be noted that the lift force may act upwards or downwards.

To understand the mechanism by which lift is generated by a hydrofoil, and evoke the idea of irrotational flow (i.e. inviscid flow, non viscous) consider the flow of an incompressible fluid past a cylinder of infinite length, so the flow can be considered twodimensional. The flow is symmetric about both the x, and y-axes, therefore no net force exists on the cylinder. If a clockwise circulatory flow or a vortex is now superimposed around the cylinder, by convention such a flow is considered negative. The result is that the symmetry of the flow about the x-axis is now disturbed. The fluid flows faster around the upper half of the cylinder compared to the lower half, resulting in the lower pressures

$$\operatorname{Re} = \frac{\mu + \mu}{\mu} = \frac{\mu}{\nu}$$
(2.1)

above, and higher pressures below (from Bernoulli's equation). This pressure difference generates an upward force parallel to y-axis (i.e. lift). There are of course no viscous forces since the flow is inviscid.



Figure 2.2 Uniform flow past a cylinder, with circulation (adapted from [1]).

Stagnation points are present at the points denoted by S_1 , and S_2 , at these locations the fluid is brought to rest. Mathematically the flow pattern may be described by the stream function ψ , constant values of which describe the geometry of the streamlines. The superposition of a two dimensional doublet on the uniform flow. Uniform flow past a circular cylinder, with a circulation Γ , yields the stream function

$$\psi = -U_{\infty} \left(r - \frac{a^2}{r} \right) \sin \theta - \Gamma \left\{ \frac{\ln \left(\frac{r}{r_o} \right)}{2\pi} \right\}$$
(2.2)

It may be shown that the lift per unit length L in a fluid of density ρ is given by

$$L = -\rho \Gamma U_{m} \tag{2.3}$$

and is known as the Kutta-Joukowski law. When Γ is negative L is positive i.e. upward lift.

Joukowski showed using conformal mapping that the flow past a cylinder could be transformed into a more useful mathematically related shape, which could yield a lift force. By successive transformations, the flow around a cylinder the flow around a hydrofoil can be obtained. The result may look similar to that shown in figure 2.2, in which the stagnation points S_1 and S_2 correspond in the transformed flow to those in figure 2.3. A sharp trailing edge is obtained from the transformation, but the required instantaneous in flow direction round the trailing edge towards S_2 cannot be sustained by an inviscid fluid. This even less likely in an viscous flow where the energy of the flow is



Figure 2.3 Uniform flow past an airfoil, with circulation (adapted from [1]).

Continuously reduced by shear stresses at the under the surface of the hydrofoil. Consequently, the only stable position at the trailing edge.

Unlike the cylinder, where the magnitude of the circulation can varied arbitrarily, the presence of the sharp trailing edge prescribes a unique value of circulation for the hydrofoil. Since there are many different foil shapes and orientations, the value of the circulation required to move the aft stagnation point to the trailing edge would vary. It follows the value of the circulation will depend on foil shape and orientation (i.e. angle of attack). Equation 2.3 states that the lift generated by an airfoil is directly proportional to the circulation about it. Hence, lift is a function of foil cross section, and its angle of attack. In general, the lift increases with increase in angle of attack, for small angles of attack, the relationship between lift and angle of attack is linear, as shown in figure 2.4.



Angle of attack, α

Figure 2.4 Lift versus angle of attack.

Viscous effects of the flow forces should also be considered when evaluating the forces exerted on a body by a fluid. Theses effects account for the forces exerted in the direction of the flow (drag) and for the deviation of the lift from the linear behaviour shown figure 2.4 due to flow separation.

Chapter 3

Design methodology and tools

3.1 Introduction

Modern design has come a long way in the last decade due to many factors including changing manufacturing techniques, new materials and changed technologies. Products have become far more complex and the market has required far more rapid development from concept to market. The aim of this chapter is to look at the influence the development of computing has had on design, and the design process and look at how computers are used in the operation.

In terms of design, the computer has become is a machine which assists with the task of product design and manufacture, processing information concerned with the product definition and operations to be performed on it. Examples of this include CAD (computer aided design) and CAM (computer aided manufacture) however there are many other tools associated with it for design analysis and simulation, which include FEA (finite element analysis) and CFD (computational fluid dynamics). In many cases these packages will allow a design to be optimised for certain criteria such as stress distribution or minimum weight, which affect the shape or form of the resulting design. ¹

3.2 Design Methodology

Engineering design can be broadly defined as the process of devising, a component, process or a system in response to a need. It is essentially an exercise in applied creativity. In this process basic sciences, mathematics, engineering sciences, and past experience are used to convert available resources optimally to satisfy a stated objective. The essential steps of any design process are as follow:

(i) Identification of need.

(ii) Background research to understand the problem.

(iii) Task specification.

(iv) Synthesis or brainstorming.

(v) Analysis.

(vi) Selection.

(vii) Construction.

(viii) Testing.(ix) Evaluation.(x) Production.

The above list may give a misleading impression that the process can be accomplished in a linear fashion as shown. More often than not iteration is required within the entire process, as I discovered for myself, moving from any step back to the previous step. For example, the ideas generated during the synthesis or brainstorming step may be found to be flawed when analyzed. Thus, a return to the synthesis stage to think alternatives or even a return to the background research may be necessary to gather more information. In general, one cannot design in a linear fashion. It is two steps forward and one (or more) back. Theoretically, this iterative process on a design problem could continue indefinitely, continually creating small improvements, but at some point, you must declare the design acceptable.

3.2 Pro/Engineer

Nowadays a number of design-related tasks may be preformed with the assistance of a computer. These tasks include:

- (i) Interactive computer graphics.
- (ii) Computer-aided analysis.
- (iii) Computer aided draughting.
- (iv) Computer aided manufacturing.

The advantages of using a computer to aid the design and manufacturing process are enormous. They include improved engineering productivity, shorter lead time, reduced labour, improved accuracy, savings in materials, faster machining times, faster response to changes in designs, improved documentation.

All three-dimensional models of the different components which make up the new improved U-Kite had been developed using PTC's¹ Pro/Engineer solid modeling program

¹ Parametric Technology Corporation

suite. A number of Pro/Engineer programs or modules were used to perform different tasks e.g. part and assembly design, model creation, production of engineering drawings, and NC machining.

Essentially Pro/Engineer is a parametric; feature based solid modeling system. "Feature based" refers to the fact that parts and assemblies can be created by defining features such as extrusions, sweeps cuts, holes, slots blends and so on, rather than specifying lowlevel geometry entities such as points, lines, arcs and so on. This means the designer can leave all low-level geometric detail to Pro/Engineer to figure out. Features are specified by setting values and attributes of elements such as reference planes, or surfaces, direction of creation, pattern parameters, shape, dimensions, and others. "Parametric" means that the physical shape of a part or assembly is driven by values assigned to the attributes (primarily dimensions). The designer may define or modify a feature's dimensions or other attributes at any time during the design process. Any changes will automatically propagate through a model. The designer can also relate the attributes of one feature to the attributes of another. For example, if a designer's intention is create a block with a hole at its centre, the designer can relate the dimensional location of the hole of the hole to the block dimensions using a numeric formula, so if the dimensions of the block are changed the centred hole position will be automatically recomputed. Parametric modelling is most efficient in working with designs where changes are likely to consist of dimensional changes rather than grossly different geometries.

Once mastered solid modeling is a far more natural mode of expression than engineering draughting. It can be used at many different stages in the design cycle and manufacture of the product. At the conceptual design stage, it can provide a visual aid, making it unnecessary to fabricate a prototype. Solid modeling allows exploded views, and sectional views at any position or orientation. The three dimensional geometry can also be transferred so that structural, thermal, and motion analysis may be preformed or used to generate numerically controlled machining data. Geometry related quantities might also be determined, volumes, weight, centre of gravity, mass moments of inertia, products of inertia, axial radii of gyration, the principal axes to name a few.

Solid modeling also allows the designer to perform interference checks, i.e. design errors, which could be potentially disastrous at the machining or assembly can be avoided by on screen simulation of the machining operation or assembly.

1 7 JUN 2003 PALWAL 121956 LIBRARY

Chapter 4

Redesigning the U-Kite

4.1 Design brief

Despite having being successfully deployed towed and retrieved on several occasions. The upper wing of the initial U-Kite prototype eventually failed during the retrieval stage of a sea trial. In view of the above, it was believed that the structure of original unit needed to be redesigned to ensure that it could support the forces it is likely to encounter during use. The type of forces which the vehicle is likely to encounter include:

- (i) Surface loads, which act on the surface of the vehicle i.e. hydrodynamic and hydrostatic pressure.
- (ii) Concentrated shock loads, for example loads encountered during transportation from the ships deck to the water, and vice versa.
- (iii) Body loads, which act over the volume of the structure, and are produced by gravitational, buoyancy, and inertial effects.

An assessment of the design of the initial prototype revealed a number of shortcomings. Most notably

- (i) The length of the main foils relative to the their chord lengths (i.e. aspect ratio) were too large at 4.2:1. Using low aspect ratio foils would result in reduced bending moments at foil roots, and a more rigid, compact, and less cumbersome vehicle. In addition, the usable range of angles of attack is wider (i.e. stall is postponed even up to 45°) and ocean currents would have less influence.
- (ii) The angles of attack of the main foils were fixed and could not be altered, thereby limiting the towing speed to a very narrow range, and limiting the vehicle's potential usefulness.
- (iii) The unit was heavier than water this meant that in the event of the cable breaking the U-Kite and it's payload of valuable instrumentation would sink, and almost certainly be lost.
- (iv) Its outward appearance would certainly need to be improved if it was to ever attract customers.
- (v) The space used to mount instrumentation was very limited and, would need to be increased to accommodate a wider range of instrumentation.

4.2 Vehicle design

The overall drive on the design of the new vehicle was:

1. To provide a simple, yet efficient towed underwater apparatus capable of achieving and maintaining a controlled depth, possessing a high degree of stability, and which may be manufactured inexpensively and does not require skilled personnel to operate.

2. To provide a towed underwater vehicle that has improved structural configuration.

3. To provide a towed underwater device, which develops a high outward lift force relative to its size and buoyancy in water and which is capable of seeking and maintaining a predetermined depth and towing orientation.

4. To provide a towed underwater device having lift generating surfaces (i.e. wings), the angle of attack of which may be adjusted manually, by an operator prior to deployment, so as to cause the vehicle to run at a predetermined depth, and distance from towing vehicle.

5. To provide a towed underwater apparatus, which is highly stable over a wide range of speeds. Which is capable of assuming a predetermined orientation regardless of its initial position.

6.To provide a towed underwater apparatus, which includes means for stabilizing the apparatus with respect to undesirable rolling, yawing and pitching actions and, means for enabling the vehicle to maintain a predetermined depth without undue stress on the towing apparatus.

7. To provide a towed underwater apparatus which is aesthetically pleasing. From an engineering point of view this may not seem very important but it generally plays an important part in the commercial success of most products.

4.3 Design constraints

The following design constraints were imposed on the design of the new U-Kite.

- (i) The overall length of the main body or the hull should not exceed one metre in length, a compact, portable design is desirable, this will allow a single person to deploy and operate the system from a very small boat.
- (ii) The main body should have a circular cross sectional with a diameter of a least 120mm. Since most commercially available oceanographic instruments which are mounted on towed vehicles come enclosed in cylindrical water-resistant housings. Such housings are typically 100mm in diameter (see figures 4.1 and 4.2).
- (iii) The hydrofoils should have a low aspect ratio to keep bending moments at foil root to a minimum; they should also have a symmetrical section (NACA 4 digit section) this will ensure they are easy to manufacture.
- (iv) The vehicle assembly should be positively buoyant in water, making retreival of the system easy at the end of survey operations.
- (v) The U-kite should easy to assemble and disassemble, so it can be broken down into packages easily manageable for transportation.
- (vi) The vehicle should be economical to manufacture. (i.e. requiring little specialist equipment, and skills)
- (vii) The vehicle should be fabricated of lightweight yet strong materials, the system should be neutrally buoyant, capable of withstanding any unexpected shock loading.
- (viii) The assembly should be corrosion resistant, since it will be operating in a very corrosive environment.
- (ix) The vehicle should be capable at operating at speeds of up 10knots (5.144 metres per second) and at depths of up to 20 metres.



Figure 4.1 Minitracka, flourimeter for chlorophyll-a, dye tracing, or turbidity profiling.



Figure 4.2 Aquatrace, flourimeter, for chlorophyll-a, and other flourophor detection, rhodamine and flourescin dye tracing, turbidity profiling, measuring dissolved oxygen, temperature, and pH.

4.4 New Design

As an alternative to the previous design, a U-Kite with the same hydrofoil arrangement (i.e. with a dihedral¹ of 40°) was proposed. The vehicle framework of the new design is fabricated from aluminium alloy, while the hydrodynamic profile is provided by removable fibre reinforced plastic fairings. This design utilises foils with an aspect ratio of 2.1, and a hull with a diameter of 140mm. Because the original u-kite actually preformed quite well when being towed, it was decided that the new foils should have roughly the same surface area as the foils on the original U-Kite. Other critical dimensions were also preserved for example the volume of water displaced by the pods at the ends of the foils (i.e. 2.5 litres). The details of the redesigned hydrofoil are given in figures 4.3 to 4.7.

¹ Dihedral refers to the angle formed by two intersecting planes, i.e. the angle between the upper and lower foils, δ -180°, see figure 4.18.



Figure 4.3 The U-Kite



Figure 4.4 U-Kites with fibre reinforced fairings removed

Probably the most obvious difference between the new and old designs is the shape of the pod at the ends of the foils. A tear shaped pod was adopted in the new design primarily to

reduce the area of the pod exposed to the turbulent boundary layer, and thus a significant reduction in drag is expected. The profile of the foils uses the NACA0021 section (see appendix A1). This profile should be sufficiently deep to provide enough space for the longitudinal structural members. These members will support resist bending moments and axial loads. They also provide a means by which the hydrofoils are secured to the main body of the U-Kite.



Figure 4.6 Cross sectional view of U-Kite.

Other changes to the design of the U-Kite include the use of a torpedo shaped hull the details of which are illustrated in figure 4.7



Figure 4.7 Torpedo shaped hull.

The hull design comprises of a streamlined body beginning with a hemispherical nose portion for maximising the payload capacity of the hull and providing a streamlined profile in the water. The middle portion consists of a cylindrical section with two flatmachined surfaces onto which the hydrofoils are secured. The hole and the circular slot permit the operator to position the foils at the desired angle of incidence, before the bolting them rigidly into position. The tailplanes are simply constructed from two flat plates of glass reinforced nylon, the purpose of these plates is to resist the pitching, rolling and yawing moments the vehicle may experience while being towed.

4.5 Lifting line theory

The first step in the design process was to determine the loads acting on the vehicle. The most important of these is the lift generated by foils. The forces on the wings are related to the properties of the airfoils:

Defining the area of each of the foils as A, the lift coefficient C_L may be defined as

$$Lift = \frac{1}{2} \times C_L \times A \times \rho \times U^2$$
(4.1)

$$C_{\rm L} = \frac{dC_{\rm L}}{d\alpha} \times \alpha = \text{Coefficient of lift, and } \alpha = \text{angle of attack}$$
(4.2)

 C_L is controlled by changing the incidence angle α .

$$C_{L} = \frac{dC_{L}}{d\alpha} \times \alpha = \text{Coefficient of lift.}$$
(4.3)



Figure 4.8 Effect of aspect ratio on lift coefficient of a typical finite span, symmetric section hydrofoil (adapted from [1]).

The distribution of hydrodynamic load on a foil of finite span in the simple, classical case of inviscid flow, may be determined using lifting wing theory. Further references regarding lifting line theory and the underlying principles discussed in this section can be found in [1,12,13].

When a hydrofoil is accelerated from rest, at the instant of starting the flow is a potential flow without circulation and, the streamlines are as shown in figure 4.9a with a stagnation point occurring on the rear upper surface of the hydrofoil. At the sharp trailing edge, the water is required to change direction suddenly. However because of viscosity (i.e. the large velocity gradients produce large viscous forces) the water is unable to flow around the trailing edge. Instead, a surface discontinuity emanating from the form the trailing edge is rolled up into a vortex, which is called the starting vortex. The stagnation point moves towards the trailing edge, as the circulation around the hydrofoil, increases in intensity, the lift increases until the flows from the upper and lower surfaces join smoothly at the trailing edge, as shown in figure 4.9d. Thus, the generation of circulation around the hydrofoil, and the resultant lift are necessarily accompanied by a starting vortex, which results because of the effects of viscosity.


Figure 4.9 The generation of circulation around the hydrofoil, and the resultant lift (adapted from [16]).

lifting line theory, the wing is replaced by a straight line and the circulation about the wing is replaced by a vortex filament. This vortex filament lies along the straight line, and at each spanwise station, the strength of the vortex is proportional to the intensity of the local lift, a suitable distribution of vortices can be used to represent the physical wing, the portions of vortices lying along the wing are called bound vortices, the bound vortex cannot end abruptly at the tips of the wing, a vortex cannot end in a fluid only at a solid surface, but joins with the vortices generated at the tips, and which trail down stream indefinitely. Flow near the ends takes place around the tips from the high to the low pressure regions similar vortices are generated along each half span at the trailing edge, which subsequently roll up to form a pair of contrarotating vortices, and which trail down ward component of velocity at the trailing edge of the wing. The effect of downwash is to change the relative direction to of the flow stream over the wing. The rotation of the flow effectively reduces the angle of attack of the wing.



Figure 4.10 Trailing vortex system(adapted from [12]).

At the tips the pressure difference between the upper and lower surfaces of the wing is zero, so at these locations the lift force and circulation is zero, maximum values of lift force occur at the mid-span location (see figure 4.12). The lift coefficient varies along the wing and so the system can be modeled as a superposition of a series of bound vortices of different strengths.



Figure 4.11 Generation of the trailing vortices, view from the trailing edge

The spanwise circulation distribution is represented by a Fourier sine series consisting of N terms:

$$\Gamma(\phi) = 4sU_{\infty}\sum_{n=1}^{N}A_{n}\sin n\phi \qquad (4.4)$$

The spanwise coordinate (y) has been replaced the f coordinate: $\frac{y}{s} = -\cos\phi$ (4.5)



Figure 4.12 Spanwise circulation distribution.



Figure 4.13 Geometry for the calculation of induced velocity at $y = y_1$.



Figure 4.14 Induced flow (adapted from [12]).

Since the spanwise lift distribution represented by the circulation of figure 4.11, the section lift force [i.e., the lift acting on that spanwise section] is given by:

$$\Gamma(\phi) \qquad \qquad l(\phi) = \rho_{\infty} U_{\infty}^{2} s \sum_{n=1}^{N} A_{n} \sin n\phi \qquad (4.6)$$

To evaluate the coefficients $A_1, A_2, A_3, \dots, A_n$, it is necessary to determine the circulation at N spanwise locations. Once this has been done, the resultant N linear equations may be solved for the coefficients.

Section lift coefficient is defined as:

$$C_{l}(\phi) = \frac{Lift_per_unit_span}{\frac{1}{2}\rho_{\infty}U_{\infty}^{2}c}$$
(4.7)

Using local circulation to determine the local lift per span, we obtain

$$C_{t}(\phi) = \frac{\rho_{\infty}U_{\infty}\Gamma(\phi)}{\frac{1}{2}\rho_{\infty}U_{\infty}^{2}c} = \frac{2\Gamma(\phi)}{U_{\infty}c}$$
(4.8)

Letting the equivalent lift-curve slope $\left(\frac{dC_l}{d\alpha}\right)_e$ = ae, and noting that the effective angle of attack $\alpha_e = \alpha - \varepsilon$ (α_{0l} is angle of attack where the coefficient of lift is equal to zero, α is the angle of attack, & ε is the downwash angle, downwash has the effect of tilting the undisturbed fluid),

$$C_{l} = \left(\frac{dC_{l}}{d\alpha}\right)_{e} \left(\alpha_{e} - \alpha_{0l}\right)$$
(4.9)

Five parameters in the previous equation depend upon the spanwise coordinate f these include:

(i) Γ , the local circulation.

(ii) c, or chord length, which may vary with f for a tapered wing

- (iii) α , the local geometric angle of attack, which varies if the wing is twisted.
- (iv) α_{0l} , the zero lift angle, which may vary if the airfoil section changes with f,

i.e. aerodynamic twist

(v) ε , angle of downwash, which dependent on the circulation distribution.



Figure 4.15 Trailing vortex system.

Using Mathematica to perform the necessary calculations, the foil characteristics may be easily determined. . In the Mathematica notebook (Apendix D), a four term Fourier series

is used to represent the spanwise loading, on a rectangular hydrofoil with a chord length of 160mm, and span of 336mm, disposed at an angle of attack of -4° . The technique can only provide a rough estimate of the real coefficient of lift since it fails to take in to account the presence of the hull and a pod at the end of the foil (both of which limit vortex shedding), it is almost certainly likely that the real coefficient of lift would be greater.

4.6 Static stability

The forces acting on a fully submerged U-Kite moving forward at a steady speed are shown in the free-body diagram figure 4.18.



Figure 4.18 Forces acting on U-Kite while being towed.

Where

- b = buoyancy of upper float
- w = weight of the ballast
- l = force on upper wing with angle of attack $\alpha_{\rm U}$
- k = force on lower wing with angle of attack α_L
- γ = angle of the upper wing to the vertical
- β = angle of the lower wing from the vertical

 δ = angle between upper and lower wings (i.e. 180° – dihedral)

 θ = angle of towing cable to the horizontal

L = length of the upper and lower wings

R = radius of hull

T = tension in the towing cable

cp = the distance from the central axis of the hull to the location centre of pressure on the upper and lower wings.

U = speed of the U-Kite through the water

 ρ = density of water = 1025 kg/m³

 A_U = area of upper wing

 A_L = area of lower wing

 C_{LU} = co-efficient of lift of upper wing

 C_{LL} = co-efficient of lift of lower wing

Horizontal forces:

$$T \times Cos(\theta) - I \times Cos(\gamma) - k \times Cos(\beta) = 0$$
(4.10)

Vertical forces:

$$b - w + T \times Sin(\theta) - l \times Sin(\gamma) + k \times Sin(\beta) = 0$$
(4.11)

Thus equilibrium of moments about O gives: $l \times cp \times Cos(\gamma) - k \times cp \times Cos(\beta) - b \times (L + R) \times Sin(\gamma) + w \times (L + R) \times Sin(\beta) = 0$ (4.12)

Using values from the new design

b = 19.62 N w = 9.81 N L = 0.336m R = 0.070m $A_U = A_L = 0.336 \times 0.160 = 0.05376 \text{ m}^2$ (i.e. the areas of the upper and lower wings are the same) cp = 0.238 m $\alpha_L = -2^\circ$ (the angle of attack of lower foil is fixed) And assuming that

 $\gamma = 40^{\circ}$

 $\beta = 10^{\circ}$

If the U-Kite travels at a steady speed of 5m/s (approximately 10 knots)

U=5 m/s

At angle of attack of -2° , the absolute value of $C_{LU} = 0.1085$ (found using lifting line theory)

$$k = \frac{1}{2} \times 0.1085 \times 0.05376 \times 1025 \times 5^2 = 73.6 \text{ N}$$

 $0.2381\cos(40^{\circ}) - 17.79\cos(10^{\circ}) - 7.966\sin(40^{\circ}) + 3.983\sin(10^{\circ}) = 0$

 $1 \approx 124.2N$

: the angle of the upper wing requires to be $\alpha_{\rm U} \approx -3^{\circ}$.

Clearly, the longitudinal members inside the foils must be strong enough to resist bending moments, and axial stresses. It is obvious from the above that the lift generated by any of the foils is unlikely to exceed 150 N under normal operating conditions (i.e. at a towing speeds less than 10 knots).

Chapter 5

Material selection

5.1 Introduction

The U-Kite is best considered as a system, which comprises several components;

- i. Buoyancy, to give the desired buoyancy, close to neutral.
- ii. Ballast, to give the vehicle a low centre of gravity, and hence greater stability.
- iii. Wings or hydrofoils to generate the necessary hydrodynamic forces to pull it down to depth and move it away from the towing vessels wake and wave system, these forces result from a suitable distribution of pressure on the wetted surfaces once the u-kite is in motion.
- iv. Main body, is the enclosure which accommodates the instrumentation, as well providing the points from which the towing cable is attached to the u-kite; towing cable or tether used for towing, deploying, and retrieving the vehicle.

The selection of materials for any of the components which make up the U-kite involved choosing materials that meet performance requirements (e.g. corrosion resistance, weight, strength, etc...) with a minimum cost, while considering preliminary part design and manufacturing constraints.

5.2 Buoyancy

Net buoyancy is provided by any part of the U-Kite, which displaces a volume of water of greater weight than, itself. This will be achieved if the overall density of the item is less than that of water. Solid metal components cannot therefore provide net buoyancy, although metal tanks containing gas are used to provide buoyancy. Most essential components of most towed instrument platforms, such as structural components and towing points etc. are negatively buoyant, although any pressure resistant housings for electronics or instrumentation, depending on their content, may well be buoyant. However, the essential components of such a vehicle are likely to be negatively buoyant, and the addition of artificial buoyancy will be required to bring the U-Kite to the near neutrally buoyant state.

Overall buoyancy is not the only important factor to be considered: the stability of the vehicle within the water is also of importance. A submerged vehicle is only stable when it's centre gravity is vertically beneath it's centre of buoyancy. The centre of gravity of an object is a widely understood concept, while the term centre of buoyancy probably deserves an explanation. Buoyancy is the centre volume of an object, or alternatively it's the centre gravity of the displaced fluid if it could retain the shape of the object displacing it.

If forces disturb the attitude or position of a stable vehicle, a restoring couple is set up by the buoyant and the gravitational forces, which tends to restore the vehicle to its original position. The distance between the centre of gravity and the centre of buoyancy is known as the metacentric height. For stability of the vehicle the metacentric height must be positive. The magnitude of the metacentric height serves as an index of stability, since the greater the distance between centres of buoyancy and gravity, the greater the restoring couple set up in the disturbed state, hence greater stability of the vehicle.

In any submersible vehicle that carries sensors stability is very important, and it is for this very reason that it is important to locate any artificial buoyancy towards the top of the vehicle, while heavy, non-buoyant components should ideally be positioned towards the bottom of the vehicle, where possible.

Artificial buoyancy can be provided in numerous ways the commonest include using closed metal tanks, spherical plastic floats of the type used in the fishing industry, low density rigid foams such polyurethane, syntactic foams etc. The choice of which means to use is usually determined by the operational depth, since the effects of pressure with increasing depth has to be taken into account. Closed tanks and spherical plastic buoys are likely to implode under pressure, and low-density foam may become compressed. Syntactic foams have developed specifically for deep-water buoyancy. These are manufactured by blending small hollow plastic spheres with rigid resins systems, so the

resultant product is impervious to water and possesses mechanical properties, which can be tailored to suit specific depths. Deepwater syntactic foams have to be extremely strong and therefore denser than foams developed specifically for use in shallow water. These foams tend to be more expensive, and less efficient than low-density non-syntactic foams. Normally syntactic and non syntactic foam buoyancy units are covered with by an outer shell of fibre glass, or thin plastic skin to provide, protection against abrasion, a smooth efficient hydrodynamic surface, and increased strength. Buoyancy units tend to be colored bright yellow or orange, so that, vehicles can be easily distinguished on the sea surface making for easy retrieval.

5.3 Aluminium and aluminium alloys

To keep the structural weight down a low density material of sufficient strength and stiffness would have to be used in the construction of the U-Kite. I considered various material systems ranging from light weight composites to stainless steel. In evaluating the pros and cons of these material systems, I decided to use aluminium for the primary structural components. The reasons behind this choice are discussed below.

The metals most commonly used today in the construction of yachts, marine structures and fittings are stainless steel and aluminium. While Stainless steel is recognised as the premium material for marine applications, where it is used for its excellent corrosion resistance, lustre, and strength. It is probably too heavy to use as a structural material, since the U-Kite is required to be neutrally buoyant.

Aluminium alloys are exemplified by the high elastic modulus and high strength, when expressed per unit density. This means that aluminium alloys have a high structural efficiency. Aluminum and aluminum alloys in general develop a protective layer (aluminum oxide) which helps to protect the material in most cases. However, if for some reason the integrity of this protective film is compromised, rapid corrosion can occur because of the highly anodic character of aluminum. In salt-marine environments (where salt deposits form easily), localized corrosion will occur at small breaks or defects in the oxide film, which might lead to the development of large pits, crevice and stress corrosion can also occur. A wide range of aluminium alloys is available mostly with small additions of one or more elements. The main alloying elements are copper (2000) series, manganese (3000), silicon (4000), magnesium (5000), magnesium and silicon (6000), and zinc (7000). Despite the relative small differences in composition, there are quiet significant differences in corrosion resistance, particularly under marine conditions. In general, 3000, 5000, and 6000 series alloys offer the best corrosion resistance in marine environments; the other alloys are not suitable for marine applications, particularly those containing copper alloys (2000 series alloys).

The corrosion resistance of aluminium alloys can be improved if anodized in a sulphuric/chromic acid electrolyte to produce a clear porous anodic film. The anodic film is then sealed in hot water, or chemical water. It is possible in this process to colour the film.

5.4 Fibre-reinforced plastics

Fibre-reinforced plastics or composites were first produced about 50 years ago. While technically the term composite can apply to any combination of individual materials, we are primarily interested in fibre that has been impregnated with a plastic matrix resin. The fibreglass boat was the first large scale application of composites, and the marine industry and in particular leisure craft industry are still the largest consumers of fibre-reinforced plastics Fibre-reinforced plastics have obvious advantages for the construction of subsea vehicles, and have been used for many years. They have many unique, and outstanding characteristics, which provide design flexibility. They are corrosion resistant (although they do require protection from water absorption), easy to fabricate, relatively low cost, and they posses a high-strength to weight ratios, dimensional stability, corrosion resistance and low tooling costs. Due to their high strength and lightweight, aerospace and high performance sporting goods utilize premium composite materials such as carbon fibre and epoxies. Additionally, corrosion-resistant composite tanks and pipes offer extended service life over metals.

43

5.5 Design Considerations

Composite product manufacturing differs from traditional manufacturing in that composite materials are built up in layers (hence the term lay-up) to shape and produce an end product, whereas in traditional production, material is removed from the basic wood, metal or other stock material to form the final shape. The difference is profound enough that it's rarely successful to manufacture a composite product using a wood or metal design. Instead, it's important to design specifically for composite materials and to consider the manufacturing process as an integral part of the design phase.

5.6 Design flexibility

In arithmetic, adding one and one makes two. In fibre-reinforced plastic, adding fibres to a resin matrix creates one material whose properties cannot be predicted by summing the properties of its components. In fact, one of the main advantages of composites is the complementary nature of the components. For example, thin glass fibres are quite strong but they are also susceptible to damage. In comparison, certain plastics are relatively weak but extremely versatile and tough. The combination of these components can create a material, which is more useful than the individual components. By carefully selecting the fibre, resin and manufacturing process, designers can tailor composites to meet final product requirements that could not be achieved using other materials. While it is this combination of matrix, fibre and manufacturing process that gives composites their superior performance, it is essential to consider these elements separately.

5.7 Fibres

Glass fibres set within the resin matrix account for the strength advantage that glassreinforced composites have over unreinforced plastics. Strong and stiff glass fibres carry loads imposed on the composite while the resin matrix spreads the load across the fibres. A wide variety of properties can be achieved through proper selection of glass type, filament diameter, sizing chemistry and fibre forms (e.g., rovings, fabrics, etc.). Overall, glass fibres exhibit excellent impact resistance, high tensile strength, and good corrosion resistance. Fibbers are silica-based glasses that contain several metal oxides, which can be tailored to create different types of glass fibres. Fibres can also be produced from carbon, boron and aramid materials. Generally, these fibres exhibit higher tensile strength and stiffness than do their glass counterparts. However, the cost of these fibres greatly exceeds that of glass fibres. Carbon, boron and aramid fibres are typically reserved for high-tech applications demanding exceptional fibre properties for which the customer is willing to pay a premium. For some applications, a hybrid composite combining an expensive fibre with glass fibre can be created to improve overall performance of the composite. Hybrid solutions augment the properties of glass fibres and cost less compared to solutions relying only on advanced fibres. Because E-glass offers good strength properties at a low cost, it accounts for more than 90 percent of all glass fibre reinforcements. Named for its good electrical resistance, E-glass is particularly well suited to applications where radio-signal transparency is desired, as in aircraft radomes and antennae. E-glass is also used extensively in computer circuit boards to provide stiffness and electrical resistance. Along with more than 50 percent silica oxide, this fibre also contains oxides of aluminum, boron and calcium, as well as other compounds.

5.8 High-strength glass fibre

When greater strength and lower weight are desired, high-strength glass or other advanced fibres, such as carbon, may be selected. High-strength glass is generally known as S-type glass. Originally developed for military applications in the 1960's.

High-strength glass has appreciably higher silica oxide, aluminum oxide and magnesium oxide content than E-glass. Typically, S glass is approximately 40 percent to 70 percent stronger than E-glass

5.9 Corrosion resistant glass fibres

While glasses are generally considered to have relatively high chemical resistance, leaching action erodes these fibres, when exposed to water. For instance, a 10-micron diameter E-glass filament typically loses .07 percent of its weight when left in hot water for 24 hours. This erosion rate slows significantly as leached glass forms a protective barrier on the outside of the filament; only 0.9 percent total weight loss occurs after seven

45

days of exposure. Because water is no friend to glass fibres, a moisture-resistant coating such as a silane compound is coated on to glass fibres during manufacturing. Resin addition during composite formation further protects the fibres.

Some types of glasses perform much better than others when exposed to acids or bases. Corrosion-resistant glass, known as C-glass, loses much less of its weight when exposed to an acid solution than does E-glass. Both C-glass and S glass show good corrosion resistance to hydrochloric and sulfuric acid. However, E-glass and S-2 glass have much better resistance to sodium carbonate solution (a base) than does C-glass.

5.10 Fibre cost

While electrical resistance, strength, corrosion resistance and thermal properties are crucial concerns in choosing an appropriate glass type; cost is often the primary factor. Bulk E-glass roving typically costs about IRP1 per pound, depending on the quantity purchased filament diameter and other factors. Corrosion-resistant roving sells for about 30 percent more than E-glass. At about IRP6 per pound, S glass roving is much more expensive than either E-glass or corrosion-resistant glass. Product designers must weigh the benefits of advanced glass fibres against their higher cost.

5.11 Forms of reinforcement

Because glass filaments are extremely fragile, they are supplied in bundles called strands rovings or yarns. Within the composites industry, there is a wide variety of terminology's describing the bundling of filaments and, in many cases, the terminology is inconsistent. In current industry usage, a strand is a collection of continuous filaments. A roving refers to a collection of untwisted strands or yarns. Finally, yarns are collections of filaments or strands that are twisted together.

Rovings, chopped fibre and fabrics are the most common materials used in composite manufacturing. Rovings are supplied on a weight basis with a specified filament diameter and yield. Either single-end (one continuous strand) or multi-end (numerous strands) rovings are available. Rovings are also used to produce a wide variety of reinforcing materials, including mats, woven fabrics, braids, knitted fabrics and hybrid fabrics. Mats and fabrics keep fibres aligned prior to resin impregnation.

5.12 Mats

Chopped strand mat is manufactured from chopped strands of glass fibre, and is probably the most commonly used glass reinforcement. The glass strands are bonded together in a random two-dimensional manner with a binder and the resulting laminate is considered, for design purposes to be isotropic. Components fabricated using chopped strand mat have volume fraction of fibres rarely exceeds 40% by weight and is frequently much less.



Figure 5.1 Chopped strand mat

Chopped strand mat is considerably cheaper than woven fabric when measured on a weight basis and is invariably used as a reinforcing material in a component except when high strength is required, in which case it would be probably replaced by glass cloth. The manufacturing technique for the production of woven glass cloth is similar to that used in the textile industry. Woven reinforcement of continuos fibres permit higher volume fractions of fibres-up to about 65% by weight. A great variety of weaves are available, the most common are:

Plain weave:

Made by interlacing yarns in an alternating over and under pattern. The pattern gives uniform strength in two directions but does drape well, and as a result its use is limited to flat or slight contour surfaces.



Figure 5.2 Plain woven fabric.

Satin weave:

Typically available in four harness satin (crowfoot) or eight harness satin. In these weave styles, one yarn is carried over several yarns before going under a single yarn. The fabric is more pliable and can comply with complex contours (i.e. doubly curve surfaces).



Figure 5.3 Satin woven fabric.

Thinner fabrics produce laminates of high tensile strength but lower interlaminar cohesion and on a weight basis are expensive than heavier fabrics. Woven rovings are used in mouldings and laminates to produce high directional strength characteristics.



Figure 5.4 Woven roving.

Unidirectional roving cloths have great strength in one direction; this is accomplished by using a high percentage of roving for the warp direction, and a small percentage in the weft direction. Bi-directional roving cloth has high strength properties in two directions at right angles to each other.

5.13 Resins

Matrix resins for glass-reinforced composites bind the reinforcing fibres together and to a certain extent protect the fibres from impact and environmental assault. In continuously reinforced composites, fibre properties, particularly strength will dominate; when glass is used as a discontinuous reinforcement, resin properties will dominate the material, with the glass enhancing properties for which the resin has been formulated.

The variety of polymer matrix resins available to engineers for designing and fabricating glass reinforced composite parts is substantial. They generally fall into two categories thermoset and thermoplastic. The difference originates in polymer chemistry and manifests in processing and final form. While both resin types are chemically comprised of molecular chains, thermoset chains crosslink during the cure reaction (set off by heat, catalyst, or both) and "set" into a final rigid form. This is not the case for thermoplastic molecular chains, which are processed at higher temperatures and remain "plastic," or capable of being reheated and reshaped more than once. Thermoplastic resins are generally used very large-scale production, and thus were not considered.

5.14 Thermosets

Among continuously reinforced glass fibre plastic parts, thermoset plastics have established themselves as the prevailing matrix material. As a result, many designers and engineers have more confidence in using thermosets. Thanks to a relatively low price tag, ease of handling and a good balance of mechanical, electrical and chemical resistant properties, unsaturated polyester resins have long been the work-horse in glass-reinforced composites.

The cure of thermoset polyesters is exothermic, in that the crosslinking process releases heat as it occurs. Through a careful balance of inhibitors, catalysts and accelerators, fabricators can control this cure profile in terms of shelf life, pot life, gel time, cure temperature and viscosity. A polyester formulation might have months of stable shelf life but readily catalyzes to cure within a minute. Special formulations of polyester resins can be used to improve impact and abrasion resistance as well as surface appearance. Known as gel coats, these resins are applied to a mold surface and gelled before lay-up of reinforcements and matrix resins.

Epoxy resins are also considered a workhorse thermoset. They are used primarily in structural aerospace applications — particularly with carbon fibres — and in electronics for printed circuit boards. With glass fibres, epoxies are a more expensive matrix than polyesters, but generally offer less shrinkage and higher strength/stiffness properties at moderate temperatures. Other advantages to epoxy resin systems include excellent corrosion resistance to solvents and alkalis, and some acids. As with polyesters, epoxy resins can be used in most composite manufacturing processes. They can be formulated to provide various properties and optimize the manufacturing process.

5.15 Fillers

Fillers, such as coloidal silica, gypsum, calcium carbonate (limestone), kaolin (clay) and alumina trihydrate are often used in composites to enhance performance and to reduce costs. Relative to resin and reinforcements, fillers are inexpensive. Depending on the material selected, fillers can impart certain properties, such as improved smoke and fire resistance, mechanical strength, water resistance and surface smoothness, to name a few.

5.16 Laminate design

Composite designers choose from a variety of fibrous reinforcements and resin systems to develop the part laminate. The laminate design of the part, often referred to as the fiber architecture, calls out the ply schedule, or arrangement of individual plies, to be used in the laminate lay-up. Each ply is designed to impart specific mechanical and physical characteristics to the final product. Since each fibre, and resin material brings its own contribution to the laminate, knowledge of raw material properties is the first step in designing a satisfactory composite product. Cost is also a major design consideration. An overdesigned part cannot compete with established material systems. A well-designed part, using the right materials and process to meet the application requirements, is usually commercially competitive, especially when installation and maintenance costs are factored into the total cost.

Basically, the reinforcement provides mechanical properties such as stiffness, tension and impact strength and the resin system (matrix) provides physical properties including resistance to fire, weather, ultraviolet light and corrosive chemicals.

Three characteristics must be considered when choosing reinforcements: First the fibre type (most commonly fiberglass, but also aramid and carbon); second, the form (typically roving strands, mat and fabrics) and third, the orientation, which is the basis of the fiber architecture of the part. Orientation refers to fiber direction in the part — typically either parallel (Uni/longitudinal 0°), circumferential (Bi-axial/90°) or helical (Biased/commonly $\pm 33^{\circ}$ to 45°) length of the part, and/or random continuous strands. Fiber direction can be varied enough to produce a virtually isotropic laminate with equal strength in all directions. The fiber volume (glass: resin ratio) in a part is also an important design factor: a higher fiber content makes a stronger part and also a lighter weight part since resin is heavier than glass.

Again, the resin type (polyester, vinyl ester, epoxy) and form (wet lay-up or prepreg) must be carefully chosen to ensure a successful design. Formulators can modify the basic resin type with specific chemicals and fillers to satisfy a wide range of product performance requirements. Resin viscosity usually expressed in centipoise (cps) units, is also important to achieve the optimum resin flow rate for a specific manufacturing process.

The laminate design, part size and complexity, along with the basic issues of economics, volume, production speed and market conditions determine whether the part will be built in open or closed molds, by compression molding or by one of the automated systems. To produce a strong and durable laminate by any process, the resin must thoroughly saturate the reinforcements, and the wet laminate must be compacted to remove excess resin and entrapped air.

5.17 Hand lay-up

The hand lay up method is the most commonly used method of fabricating at room temperature fibre reinforced parts. Its major advantages, is that is a very simple process so that little specialist equipment is required. To produce a part with this process, a release agent is first applied to the open mould to prevent the moulding sticking to it. A layer of gel-coat then follows this. This gel-coat has a number of functions. Firstly it conceals the irregular mesh pattern of the fibres, and secondly it protects the reinforcement from attack by moisture, which would tend to break down the fibre resin interface. Fibre reinforcement, which is normally in form of a cloth or mat, is manually placed in the mould. The base resin mixed is with catalysts and accelerators, and then applied by pouring, brushing, or spraying. Rollers or squeegees are used to thoroughly wet the reinforcement with the resin, and to remove entrapped air. To increase the wall thickness alternate layers of fibre reinforcement and resin is applied to the mould until the desired thickness is achieved. The basic laminating procedure is illustrated in figure 5.6. The finished moulding must usually be trimmed with saw to size the edges.



Figure 5.5 Illustration of the lay-up sequence commonly used in wet hand lay-up.



Figure 5.6 Lay-up procedure used in wet hand lay-up.

The following section describes the step by step procedure used to fabricate FRP¹ components during this project:

- Inspect the surface of the mould for accuracy and/or surface defects. Keeping in mind that the laminate will be a mirror image of the surface (i.e. every surface imperfection will be reproduced). All surface defects or imperfections should be repaired before proceeding to the next step.
- 2. Clean the moulding surface.
- Prepare the moulding surface with the appropriate sealers, and mould release agents (normally wax), polish mould surface to ensure easy separation of moulded components from moulds.
- 4. Prepare gel-coat by adding filler (e.g. colloidal silica, graphite, aluminium powder, wood flour, etc.) to a small quantity of laminating resin, continue to add until the mixture it sufficiently viscous so that is capable of adhering to vertical surfaces without running off (i.e. thixotropic). This mixture should have a homogenous consistency.
- 5. Add pigment to gel-coat resin.
- 6. The gel coat resin and hardener should be carefully weighted and mixed thoroughly.
- 7. The gel-coat may be applied using a brush, squeegee, or sprayer. The gel-coat should be allowed to cure until it becomes "tack free". The surface coat is considered tack

¹ Fibre reinforced plastic

free when your will leave a fingerprint in the material, but the material will not stick to your finger. When using Pro-Set epoxy 125resin and 226 hardener this takes approximately 3 hours at room temperature.

- 8. Mix laminating resin and hardener in the correct ratios, (i.e. according to the manufacturer's instructions). Never mix more resin than it is possible to use in 30 minutes. Because laminating is a time consuming operation, laminating resins have a working life of a minimum 30minutes.
- 9. Apply catalyzed resin to moulding surface, and when the surface is completely coated with resin, lay fibre mat or fabric, directly on to the resin. Once the fibre is in position, a roller is used to consolidate the laminate, and remove any trapped air. When consolidating the resin and cloth, start at the middle and work the air out to the edge. If excess resin appears, add another layer of cloth without adding more resin.
- 10. Laminating resin is again applied, and the operation described in step no. 7 is repeated.
- 11. Repeat step no. 8 until the desired thickness has been achieved.
- 12. Allow part to cure, maintain room temperature above 18°C, and use artificial heat source if necessary.
- 13. After 24 hours or so the moulding will have cured to a brittle "B" stage, and can be removed from mould.
- 14. Temper at room temperature for two weeks.
- 15. Trim moulding. This stage requires a lot of patience and elbow grease. Excess material can be removed using a coping saw (because of the nature of glass fibre the saw blades do tend to wear down surprisingly fast), all edges then can be finished using a fine file or Emery paper. When using a file move the file along the edge, not transversely, to avoid delamination.

Chapter 6

Mould fabrication

6.1 Introduction.

This chapter describes the fabrication of tooling used to mould the U-Kite's hydrofoil fairing. The fairing is a big part, and any sort of traditional tooling would be much too expensive. This project had a limited budget and therefore, everything had to be cheap. Fortunately, the design requirements all fit in with the cheap approach. The same approach was used to mould the other GRP components used on the U-Kite.

The tooling/mould is the object that maintains the liquid reinforcement--matrix mass in a predetermined shape until a dimensionally stable composite is obtained. There are many different ways in which a mould of a given shape can be fabricated; perhaps the most obvious way to fabricate a mould is to machine a solid block of material into the desired shape. A common approach is to use a computer controlled milling to machine out a solid block of metal, according to a CAD model. Another mould fabrication technique is to take a direct impression of a master model, or pattern using castable materials such as neat polymers plaster, and foams. These materials may be reinforced to strengthen the mould shell. This technique offers a rapid and accurate means of mould fabrication.

6.2 Computer numerical control.

Controlling a machine tool by means of a program is known as numerical or NC. In a typical NC system the numerical data which is required for producing a part is stored in a part program. The part program is arranged in the form of blocks of information, where each block of data contains the numerical data required to produce one segment of the component This data will also contain information concerning cutting speeds, feeds, depth of cut. Compared to a conventional machine tool, a NC system replaces the manual actions of an operator.

In NC machine tools each axis of motion is equipped with a separate driving device which replaces the hand wheel of the conventional machine. The driving device may be a DC motor, a hydraulic actuator or a stepping motor. The NC machine tool system consists contains the controller unit and the machine tool itself. The machine controller system consists of the electronics and hardware, which read and decode the part program into mechanical actions of the machine tool. NC controllers can be can be divided into types namely open loop and closed loop control types. The former does not allow positional errors to be compensated for and is used for machines that require positional or point to point control of low accuracy. Most modern machines are based on closed loop control which compensates for the positional error on the basis of feedback from the position measuring unit, thereby ensuring the accuracy and the repeatability needed for positional and profile-following control.

In the past NC controllers used vacuum tubes and electrical relays; control over machines was realised through a hydraulic servomechanism. These types of control systems were unreliable and inaccurate. With the development of electronic technology NC controllers were constructed with digital circuitry using individual transistors and integrated circuit boards. These controllers required that that the NC program be written in special codes and stored on perforated paper taped, and fed into the controller through a tape reader.



Figure 6.1 Basic components of conventional NC control system.

57

With the introduction of read only memory (ROM) technology it became possible to incorporate a dedicated computer into a NC controller, a technique called computer numerical control (CNC). A sequence of operation instructions could be stored in ROM and could be accessed and executed by the machine control unit. The tape, either paper or magnetic strip, was no longer the means to store the NC program. A program could be stored in the memory of the controller or received from a separate computer. The program stored in the CNC controller could be also modified a convenience that older types of NC controller could never have.

Initially the preparation of a part program for NC machine tools requires a part programmer. Ideally, this part programmer should possess a good knowledge of fixture design, experience of using, and setting machine tools. In the past part programs were written manually but with the development of the CAD\CAM Systems it is possible to generate automatically NC machining programs directly from a CAD models. Such software performs the numerical control function, and permits the user to define on screen the cutter path based on the part profile, and machining operations. A cutter location file (i.e. CL or CLDATA) in ASCII format with an APT (Automatic programmed tool) syntax. Defining the sequence of the machining operations and cutter motions can be generated automatically, this programs can then be Post processed using the appropriate post-processor to create a suitable NC programs to drive an NC machine tool. Within this system the cutter path can be simulated on the screen, and if proven correct, can be downloaded directly, through an RS232 interface to the machine control unit, or alternatively the program can be stored on disk for future use. Although interactive CAD\CAM systems do reduce the time for specifying (writing) programming statements, this might not be significant in certain cases. Besides well designed machining plan should be designed in advance so that the machining steps can be defined interactively.

6.3 The principles of NC program generation based on a CAD model

A machined part can be considered as the subtraction of the solid representing the redundant material from the solid representing the stock material. The cutter motion for

cutting or profiling the part should sweep over and clean out the material occupied by the redundant material, which represent the boundary of the cutter motion. Given this volume, the cutter path can be determined, by specifying the route of the cutter motion in space. Now the cutter motion can be divided into two parts, one for the rough cut, which does not necessarily follow the exact profile of the part, and the one for finishing, which is determined by the profile or boundary surfaces of the part.



(c)



An algorithm for determining the cutter path can be designed and incorporated into a cad system. So a NC cutter path can be automatically, on the basis of solid models of the cutting tool, work-piece, and the stock material after the necessary machining specifications, starting and end points, tolerance, allowable scallop height. Additional information, such as the solids representing fixtures, also can be input into the system to avoid collisions of the tool with fixtures.

6.4 Using Pro\Manufacture

In Pro/MANUFACTURE, interactive NC programs are generated using a previously defined Pro/ENGINEER model. Pro/MANUFACTURE is designed to allow a user/programmer to select existing objects to define virtually all aspects of the desired NC tool path. When you begin a programming project, the user must create a manufacturing file that contains all of the data pertinent to the finished tool path. This includes the part, which he must assemble into the manufacturing file, along with the stock material, and the machining work cell (the exact meaning of this phrase is explained later).

The users must therefore, be aware, without the system telling him, what the stock you start with looks like. The stock can be defined by either modeling it or by assembling an existing part into manufacturing file.



Figure 6.3

The user must decide how the rough and finish tool paths must be defined to properly create the part from the stock. He can then select existing edges, faces, points, or datum curves to define the tool path.

Within NC Programming in Pro/MANUFACTURE, there are three "objects" which store data; Workcells, Operations and NC Sequences. The Workcell stores all of the information associated to the Machine Set-up; namely, the machine type (Mill, lathe,

edm, laser, etc...), the min/max axis travel of the machine, and the number or axes (e.g., 3, 4, or 5 axes) it has. It also stores references to the Work piece (stock) and the Operations that were created. You can also store a series of tool definitions with a Workcell. This is extremely useful if you normally utilize a fairly standard set of tools with a particular machine tool.

The Operation stores the NC Sequences (tool paths). The Operation also stores the Machine Coordinate System (MSYS). All output (e.g., a GOTO Point) is calculated in this coordinate system. The orientation and origin of this coordinate system must match the NC Machine.

NC Sequences store references to the geometry being machined, the clearance plane, and the machining parameters. NC Sequences also store references to the tools being used. NC Sequences also store the GOTO points. These are calculated from the existing references whenever a Sequence is generated. To create a NC program, the user must generate a series of NC Sequences. A NC Sequence is defined using the same general procedure, no matter what type of machining is being preformed. A simplified process flow for creation of typical NC Program is shown below. The basic steps required to machine in 3, 4 or 5 axis using Pro/Manufacture are almost exactly the same. The system varies in how you define the machining geometry, and in the use of machining parameters such as Step Over, Tool Description, etc.



Figure 6.4 flow for creation of typical NC Program

6.5 Pro\NC-CHECK

By using PRO\NC-CHECK for the verification of tool paths, prior to accepting the tool paths and saving the tool paths user/programmer can prove the code without having to wait until the end of the programming sequence to verify the NC programs. PRO\NC-

CHECK uses a verification technique that actually simulates the tool's removing material from the workpiece, therefore reducing the risk of accidentally gouging the part or any other geometry that user may wish to avoid such as fixtures for machining.

6.6 PRO\NC-POST

Postprocessing is the final stage that is required to convert the CL files into usable code that can drive a machine tool. Because PRO\MANUFACTURE developes CL files in a standard format, the files are not useful for driving machine tools by themselves because there are no machine tools available in the manufacturing community, and each manufacturer has it's own requirement for NC files. Because of this the information contained in the CL file cannot be used unless the information is postprocessed. Postprocessing is essentially converting the generic code into machine-specific code that is understandable to the NC machine-controller system. Because there are many variations in control systems fitted to machine tools it is necessary to have a postprocessor to suit each control system for which part programs are to be prepared. CAD\CAM software companies such as PTC (Parametric Technology Corporation) will supply specific post-processors to order, these being immediately available for the more widely used controls. The postprocessing of data into a machine control language is achieved very rapidly, being simply a case of making a few keystrokes. After postprocessing the program can be then read and executed by the NC machine-controller system. During the course of this project NC codes were generated for a FANUC controller system which was used to drive a Bridgeport Interact 408 3-axis milling machine.

The Pro/Manufacture module includes a standard set of NC post –processors that can be executed directly or modified by using the Pro/NCPOST module to suit your specific needs. To create a post-processor using Pro/NCPOST information about the NC machine and its controller must be entered into a database using the Quest facility, this information can then be used by GENER to convert the points and vectors of the cutter location file into a NC machine part program file. This file then can be punched into paper tape or transmitted to the NC machine via an RS232 link.

63



Figure 6.5 Pro/NCPOST GENER verbose window showing the postprocessing of CL files

6.7 Tooling

The prime objective of tooling is to allow a manufacturer to make identical parts of products. Depending on the complexity of a manufactured item, a single tool may be required or more frequently, several tools are required. If two or more tools are required, It is essential that they co-ordinate very accurately with one another. If proper coordination is not maintained, individual parts made on different tools will not fit together properly when placed together in an assembly.

From the proceeding it is obvious that in order to ensure that all parts of a given product fit together properly, a dimensional standard is required. A two-dimensional engineering drawing or three dimensional computer models does not serve this purposes. Since, manufacturing parts of exactly equal dimensions is impossible, due to variety of physical limitations on manufacturing processes (such as cutting conditions, hardware accuracy, software accuracy, skills of machine operators, etc.). It is important to translate the computer models into a three-dimensional representation of the part or product to be manufactured. In the tooling industry, this three-dimensional object is referred to as a master model. The master model is constructed to extremely close tolerances, and once completed, serves as a dimensional standard for subsequent tooling.

A master model is a full scale, three-dimensional object, which establishes the complete outside of a part or assembly as established by the engineering drawing. The master model is the starting point for making tools for manufacturing production parts. There are numerous ways to manufacture a master model, but most employ a block of material, which is machined to the desired shape. This block maybe high quality wood, rigid polyurethane foam, aluminium, etc. While this machining may be carried out machining by hand it is more common that a computer controlled lathe or milling machine executes the machining according to a CAD model. After machining of the master model is finished, its surface is treated in much the same way as a mould fabricated through the direct manual method.

Once the master mould is finished there is a number of ways in which the mould shell can be fabricated onto the master model. The most common way is to laminate a composite on the master model using wet hand lay-up techniques, and materials. Once the mould shell has been fabricated it must be strengthened with a back up structure. Is manufactured from the same type of material as used in the mould shell to eliminate distortions due to differences in the Coefficient of thermal expansion.

6.8 Fabrication of mould tool for hydrofoil fairing

The following section describes the procedure used to build the tooling on which part of the U-Kite's hydrofoil fairing of was laid up.

Choice of design features

1. Female mould tool

(i). We wanted the outside of the hydrofoil fairing to look good, since it is the only surface the customer is likely to see. The most visually acceptable surfaces of any moulded component are those, which are cured in contact with the tool.

(ii). Since it is far easier to make a male model of the hydrofoil than a female mould. It was decided that it would be better to build a male model of the hydrofoil first, and then to create a female tool by taking an impression or splash of the male model; using wet lay up composite.

(iii). A female hydrofoil tool could probably be produced from a single block of machined aluminium, however such a tool would have been too expensive, and too heavy.



Figure 6.6

Construction of the male model

Figures 6.6 to 6.17 illustrate the steps used to produce a male model

Part A

- 1. Create Pro\Engineer Part model.
- Create Pro\Manufacture file and define stock material using the previously defined Pro\Engineer model as a reference.
- 3. Create work cell for a three-axis Milling machine that matches the machine you are using.


Figure 6.7 Pro\Engineer part model, stock material.

4. Set up roughing NC sequence, specify tool (using short series slot drill with a large cutter diameter will ensure a large material removal rate, and thus minimizes machining time). The machining parameters (i.e. cutting feeds and speeds, free feed, step depth, stock allowances, scallop height, number of passes, tool overlap, step depth, plunge depth, coolant on/off, entry/exit parameters, and so on.).

5. Generate roughing tool path and verify using ProNC\Check.

6. Set up finishing NC sequence (i.e. profiling), machining a compound or doubly curved surface will require a tool such as a ball nose slot drill. To produce an acceptable surface finish a large tool overlap¹ must be specified in the machining parameters. This will significantly reduce the material removal rate, and increase machining time.



Figure 6.8 Finishing sequence using ball nose cutter7.Generate profiling tool path and verify using ProNC\Check8 Postprocess roughing and finishing machining sequences.

¹ Tool overlap is the amount that the tool should overlap the region machined during the previous pass. This value must be less than the tool diameter.

9. Set up NC machine tool, and using NC part program machine male form.

10. Using plaster of Paris take an impression of machined form, to create a female mould.

Using uncarded long fibre hemp as reinforcement.

11. When plaster is dry, seal the mould with shellac, or French polish

12. Using the wet lay procedure described in section 5.17 lay up part A.



Figure 6.9 Plaster mould and moulded part.

Part B

- 1. Prepare CAD model of hydrofoil section.
- 2. Define machining parameters and sequences and create NC part program.
- 3. Machine hydrofoil section.





Figure 6.10 Machine hydrofoil section

4. Using plaster of Paris take an impression of machined form to create female mould.



Figure 6.11 Make impression of machined form.

- 5. When mould is dry, seal using shellac.
- 6. Spread a thin coat of moulding release agent over the entire mould.
- 7. Using polyester resin cast hydrofoil section as shown in figure 6.12.Repeat this process until enough cast sections to form a male form shown in figure 6.13.



Figure 6.12 Casting sections.

- 8. Glue the sections together to form male, see figure 6.13a.
- 9. Take an impression of the male form using wet lay up (chopped strand mat / Epoxy).
- 10. Once the mould has cured, part B may be moulded, and trimmed.



Figure 6.13 (a) Assembled hydrofoil sections (b) hydrofoil mould (c) Trimmed moulding

Part C

1.Glue two cast hydrofoil sections together to form the part shown in the figure below.



Figure 6.14

2.Using a conventional milling machine drill three equally spaced small holes on the surface machined male form shown below. The location of these holes should coincide with the centreline of the male form. Using a scribe mark a centreline on the surface of the male form, using the holes as an aid.



4. Locate glued hydrofoil sections on the machined male form as shown below, align the glue-line of the sections with the centre-line of the male form, once in position, use oil based modeling clay to fix the two components together, and allow clay to dry.



Figure 6.16

- 5. Once the clay has dried the hydrofoil sections will be sufficiently secure, to permit the space between the two components to be filled up with modeling clay, and shaped into a fillet contour. See figure 6.15
- 6. Using plaster of Paris take an impression of the male form.
- 7. Once the plaster has set, separate the female mould, from the male form.
- 8. Seal mould with shellac
- 9. Lay up part C using the previously described procedure.



Figure 6.17 Moulding component.

Having moulded all the parts they may be glued together as shown in figure 6.6 to form master model. Once the master mould is finished mould shell can be fabricated onto the master model. Using wet hand lay-up techniques, and materials. Once the mould shell has been fabricated it must be strengthened with a back up structure.



Figure 6.18 Complete mould for one half of the hydrofoil

Chapter 7

Computational Fluid Dynamics

7.1 Introduction

In order to evaluate whether the design of the U-Kite can withstand the hydrodynamic loads it is likely to encounter while being towed, it was necessary that some analyses were conducted. In view of the complexity of the flow fields about most flight vehicles whether moving above or below water, hydrodynamicists/aerodynamicists, often develop theories for flows about components of such vehicles. However the designer is faced with the fact that the hydro/aerodynamic load on the vehicle are not simply the sum of the of the loads of the individual components (i.e. hydrofoils, hull ,etc ...); the difference is referred to as hydro/aerodynamic interaction. Wind tunnel, and tow tank tests have shown that that there is significant interaction of loads can be a significant contribution to the total loading on the vehicle. Consider the diagram shown below of the U-Kite. There are several interaction effects the effect of the foils on the body, the body on the foils. The pressure differential between the upper and lower surfaces of the foils will carry over onto the body, and generate a net lift. Other interaction effects can be also be present (e.g. foils/tail, tail/foils, cable/ foils, etc...). There are a number of numerical procedures available with which to attack this apparently insoluble problem. The finite element method is one such method.



Figure 7.1 Regions of positive and negative pressure.

7.2 Vicous incompressible fluid flow with inertia effects

A fluid is any substance, which will flow or deform under the action of applied shearing stresses. This implies that shear stresses do not exist in a fluid unless in all real fluids, a shearing deformation is accompanied by a shearing stress. The fluid of interest in this thesis is seawater which is Newtonian in nature, this means that the Shearing stress is proportional to the rate of shearing deformation. The constant of proportionality is called the coefficient of viscosity, μ . In a Newtonian fluid the relationship between stress and the rate of deformation is:

$$\tau_{ij} = -P\delta_{ij} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \delta_{ij}\lambda \frac{\partial u_i}{\partial x_i}$$
(7.1)

Where: τ_{ij} = stress tensor

P = Pressure

 $u_1 =$ Orthogonal velocities (i.e. $u_1 = V_x$, $u_2 = V_y$, $u_2 = V_z$, where V_x , V_y ,

And V_z = flow velocity components in the x, y, and z directions respectively).

 $\mu = 1^{st}$ coefficient of viscosity

 $\lambda = 2^{nd}$ coefficient of viscosity or coefficient of bulk modulus

 $\delta_{ij} = \text{Kronecker delta} (i = j \implies \delta_{ij} = 1, i \neq j \implies \delta_{ij} = 0)$

A viscous flow is one in which the fluid flow is assumed to have a non-zero viscosity, and depending on the overall dynamic behavior of the fluid flow, the flow may be a laminar (orderly state of flow in which the fluid particles move in a smooth fashion, in layers, or laminae). Turbulent (the paths of individual fluid particles are not laminar, instead the fluid particles move with a great deal of freedom, fluid particle motions have fluctuating motions, and erratic paths, mixing of laminae occurs both in the lateral to and in the direction of the main flow). Transition flow (this occurs whenever the laminar flow becomes unstable and approaches the turbulent state.

Most fluids exhibit some viscous behaviour. This causes energy dissipation that affects the motion of fluids. The governing equations of motion of incompressible are the momentum equations, the continuity equation, and the energy equation. The equation of continuity assumes the following form:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho V_x)}{\partial x} + \frac{\partial (\rho V_y)}{\partial y} + \frac{\partial (\rho V_z)}{\partial z} = 0$$
(7.2)

Where: V_x, V_y , and V_z = flow velocity components in the x, y, and z directions respectively

$$\rho$$
 = Density
 t = Time
 x, y, z = Global Cartesian coordinates

Where: P =pressure

Or using vector notation

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \rho \vec{V} = 0 \tag{7.3}$$

Where:

$$\vec{V} = V_x \vec{i} + V_y \vec{j} + V_z \vec{k} = \text{velocity vector}$$
(7.4)

$$\vec{\nabla} = \frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k} = \text{Gradient vector}$$
(7.5)

For the incompressible flow condition, the time rate of expansion in volume of a fluid element or particle will be zero and hence the continuity equation, for steady and unsteady flows becomes:

$$\vec{\nabla} \cdot \rho \vec{V} = \frac{\partial (\rho V_x)}{\partial x} + \frac{\partial (\rho V_y)}{\partial y} + \frac{\partial (\rho V_z)}{\partial z} = 0$$
(7.6)

$$\left(\vec{V}\cdot\vec{\nabla}\right)\rho + \rho\vec{\nabla}\cdot\vec{V} = V_x\frac{\partial\rho}{\partial x} + V_y\frac{\partial\rho}{\partial y} + V_z\frac{\partial\rho}{\partial z} + \rho\left(\frac{\partial(V_x)}{\partial x} + \frac{\partial(V_y)}{\partial y} + \frac{\partial(V_z)}{\partial z}\right)$$
(7.7)

$$\rho \vec{\nabla} \cdot \vec{V} = \rho \left(\frac{\partial (V_x)}{\partial x} + \frac{\partial (V_y)}{\partial y} + \frac{\partial (V_z)}{\partial z} \right) = 0$$
(7.8)

$$\vec{\nabla} \cdot \vec{V} = 0 \tag{7.9}$$

The continuity equation essentially states that mass flow rate into a specific volume must equal the mass flow rate out of that volume. This is used as a constraint on the solution to any problem, and ensures that no extra fluid is created where it could not or should not exist.

The momentum equations are:

Where: g_x, g_y , and g_z = components of acceleration due to gravity

 μ_e = Effective viscosity (see turbulence modeling)

 R_x , R_y And R_z = distributed resistances (e.g. flow through a screen, or porous media)

 T_x , T_y And T_z = Viscous loss terms (the Viscous loss terms which are eliminated in the incompressible constant property case).

$$\rho \frac{DV}{Dt} = \rho g - \nabla P + R + \nabla \tau_{ij} + T$$
(7.13)

The above equations are obtained by applying Newton's 2nd law that is:

$$\vec{F} = m \frac{d\vec{V}}{dt} \tag{7.14}$$

Where: $\frac{d\vec{V}}{dt}$ = total derivative of \vec{V}

Since $\vec{V}(x,y,z,t), x(t), y(t)$ and z(t), the total derivative is the sum of the local and convective derivatives:

$$\frac{d\vec{V}}{dt} = \frac{\partial\vec{V}}{\partial t} + \left(V_x \frac{\partial\vec{V}}{\partial x} + V_y \frac{\partial\vec{V}}{\partial y} + V_z \frac{\partial\vec{V}}{\partial z} \right)$$
(7.15)



When the local time dependent changes $\frac{\partial V}{\partial a}$ are equal to zero flows are said to be steadystate flows, because the local may be equal to zero this does not necessarily follow that $\frac{dV}{dt}$ will equal zero since, flows which are nominally steady may have large accelerations due to convective terms. Wherever convective effects occurs in the basic laws involving the mass, momentum, or energy, the basic equations become nonlinear, and are usually more complicated than flows which do not involve convective changes. The differential equations for the linear momentum $(7.10 \rightarrow 7.12)$ are known as the Navier-Stokes equations. The unknowns in the Navier-Stokes and continuity equations are the velocities V_x , V_y , V_z and the pressure P.

7.3 Finite element method

There are many ways to solve linear and non-linear boundary and initial value problems, ranging from completely analytical to completely numerical. Exact solutions are usually available for a few problems with simple domains (e.g. rectangular). These solutions are only obtainable by direct integration of the governing differential equations. These solutions can be found using such techniques as separation of variables, Laplace and Fourier transformation transforms. How ever real problems often involve geometrically complicated domains, multiple dimensions and non-linearities. Analytical solutions to such problems may not exist or may not be feasible. Approximate solutions are usually sought for such problems, using techniques such as Rayleigh-Ritz, finite difference, Boundary element, and finite element. The finite element method provides an elegant means to solve such complex problems.

The finite element method originated in the 1960s as a technique for structural analysis. Since that time, and in parallel with advances in computing power that have occurred it has developed into a powerful technique for the solution of a wide range of differential equations used in engineering and science. The finite element method is essentially an approximate method for calculating the behaviour of a real domain by performing an algebraic solution of a set of equations describing an idealised model with a finite number of variables. In the model the domain is idealized by a set of elements.

The major advantage of the method it is geometrical flexibility. It is easy to carry out solutions on domains of irregular shape, and to vary resolution within the domain to concentrate the computing effort where it is most needed.

The following steps summarize the procedure in solving a typical finite element problem. Divide the solution domain into small non-over lapping elements (e.g. triangular or quadrilateral elements for a 2 dimensional problems, or tetrahedral or hexahedral elements for 3 dimensional problems). Which are defined and located by the position of points in the domain known as nodes .The elements are joined to adjacent elements along common faces or edges where they may share nodes, in particular at element corners. The collection of nodes, and elements describing the domain is known as the mesh. Over each element the behavior is of the variables is described by the governing differential equations.

Define the behaviour of the variables in each element by a suitable shape function, these shape functions describe how the variables change over each element (e.g. linearly, or quadratically). The higher the order of the shape function, the more nodal points are assigned to each element. Accuracy of the solutions can be improved by either increasing the order of the shape function (e.g. use quadratic elements), or alternatively by using a larger number of elements with a lower order shape function (e.g. use linear elements). Apply to the mesh the boundary conditions, such as specified loads, displacements, temperature, and velocity.

Solve, the distribution of the variable of interest is approximated by the shape function, which may be described by a set of equations. The equations for all the elements in the mesh may be solved using direct (e.g. Gaussian) or indirect (Gauss-Seidel) iterations. Provided that the boundary conditions of the actual problem are satisfied then a unique solution can be obtained for the overall system of equations.

7.4 Finite element modeling

There are essentially three stages involved in applying the finite element method to any engineering problem:

(i) Idealisation of the problem

The modeler must first decide on the mesh layout, that is the number of nodes and the number elements. Regions of expected abrupt changes in the field variable (e.g. a stress concentration at around a hole) require a greater node and element density than regions where gradual changes occur. After the mesh layout has been chosen the modeler must select the type analysis (steady state/transient, linear/non-linear, etc), the type (structural, thermal, fluid, acoustic, etc.), and the number of degrees of freedom at each node, the boundary conditions, the element information (type, number of nodes, and Gauss quadrature order), material properties, constraints, and loads.

(ii) Solution, or processing of the analysis

Once the finite element model has been defined, it must be input to the code that performs the finite element analysis.

(iii) Interpretation of the results

The output from the finite element analysis codes is in numerical form. It is usually in consists of the nodal values of the field variable and its derivatives. For example in structural problems, the output is displacement and element stresses. Graphical outputs are used to allow the modeler to determine the significance of the results. It enables the modeler to interrogate the results by utilizing several types of display forms. Display analysis result contour according to specified data types, For example in structural problems; stress contours may be displayed as Von Mises equivalent stresses, principal stresses, Treasca maximum shear stresses.

7.5 Turbulence modeling

The majority of flows in nature are turbulent. Because of this fact the question is often raised whether it is necessary to include some representation of turbulence in computational models of flow processes.

7.6 Definitions and orders of magnitude

The possibility that turbulence may occur is generally measured by the flow Reynolds number,

$$\operatorname{Re} = \frac{\rho L U}{\mu} \tag{7.17}$$

Where ρ is fluid density and μ is dynamic viscosity of the fluid. The parameters L and U are a characteristic length and speed for the flow. Obviously, the choice of L and U are some what arbitrary and there may not be single values that characterize all the important features of the entire flow field. The important point to remember is that Re is meant to measure the relative importance of fluid inertia to viscous forces. When the viscous forces are negligible the Reynolds number is large.

Roughly speaking a Reynolds number well above 1000 is probably turbulent, While a Reynolds number below 100 is not. The actual value of a critical Reynolds number that

separates laminar and turbulent flow can vary widely depending on the nature of the surfaces bounding the flow and the magnitude of the perturbations in the flow. In a fully turbulent there exist a range of scales for fluctuating velocities that are often characterized as collections of different eddy structures. If L is a characteristic macroscopic length scale and l is the diameter of the smallest turbulent eddies, defined as the scale on which viscous effects are dominant, the ratio of these scales can be shown to be of the order $\frac{L}{l} \approx \operatorname{Re}^{\frac{3}{4}}$. This relation is known as the Kolmogrov length scale.

7.7 FLOTRAN

FLOTRAN is based on the finite element method, and is fully integrated with ANSYS. So that the input data for a problem can be prescribed in an interactive fashion using Ansys's pre and postprocessors. Flotran works by dividing the region of interest, the air around the outside of a car for example, into a large number of elements (the mesh or grid). In each of these elements, the partial differential equations describing the fluid flow (the Navier-Stokes equations) are rewritten as algebraic equations that relate the pressure, velocity, temperature and other variables, to the values in the neighbouring elements. These equations are then solved numerically yielding a complete picture of the flow down to the resolution of the mesh. FLOTRAN can treat two- or three-dimensional viscous flow. The formulation allows the inclusion of the energy equation when necessary, and the equation of state may be also used for variable property flows. The governing equations include the momentum equations, the continuity equation, and the equation of state. The governing equations are expressed in primitive variables i.e. pressure, velocity. In addition to the basic flow and energy equations, the two-equation turbulence model (i.e. K- ε) is available to simulate turbulent flows. This two-equation model solves two extra transport equations for turbulence (turbulent kinetic energy, K, and a length-scale/ time- in the form of dissipation scale ε).

7.8 Turbulence models

From the above relation for the range of scales it is easy to see that even for a modest Reynolds number, say $Re = 10^4$, the range spans three orders of magnitude, $\frac{L}{l} = 10^3$. In this case the number of control volumes needed to resolve all the eddies in a three dimensional computation would be greater than 10^9 . Numbers of this size is well beyond current computational capabilities. The forming and disappearing of eddies instanteously and randomly in small scale, would require very small time steps and extremely fine meshes to model the full time dependent, small scale vortices of the Navier-Stoke's equations. For this reason considerable effort has been devoted to the construction of approximate models, for turbulence. The most successful computational models for practical engineering purposes are those involving two or more transport equations. A minimum of two equations is desirable because it takes two quantities to characterize the length and time scales of turbulent processes.

7.9 Fluctuations and time-smoothed quantities

When viscous fluids move slowly around small bodies or through small channels, the flow is laminar, even when unsteady, but when the flow velocity of a fluid is increased enough, the flow will become more and more random, particularly near solid surfaces. Any flow can be characterized by its Reynolds number Re. What can be observed empirically is that the flow becomes turbulent whenever the Reynolds number exceeds a certain value (Re)tr, called the transition Reynolds number. By speeding up the flow sufficiently, the flow Reynolds number can be increased to the point where it exceeds the value of the transition Reynolds number for that flow geometry, and the flow changes from laminar to turbulent.

A classical example of transition to turbulent flow occurs in a pipe flow when the volumetric flow rate is gradually increased from zero. For low Reynolds number values, the flow is steady and laminar. However, as the flow approaches the transition Reynolds number, which has approximately a value of $(Re_{pipe})tr = 2,300$ (the absolute value depends extremely on the experimental setup), the flow begins to become unsteady,

although it is still laminar. As the Reynolds number is further increased, bursts of turbulent flow are encountered until, at higher Reynolds numbers, the flow is entirely turbulent. Thus there is a range of Reynolds numbers, centered about the transition value, over which the flow changes from completely steady laminar flow to a completely turbulent flow. Transition is not an instantaneous process. Based on the conservation equation of momentum $(7.10 \rightarrow 7.12)$, a statistical approach is used and the equation is averaged over a time scale which is long compared with that of the turbulent motion. The resulting equation describes then the distribution of mean velocity.

The instantaneous velocity V is an irregularly oscillating function and is separated into mean and fluctuating quantities. The mean velocity is used to model the large scale flow characteristics and is defined by taking the time average of V over a time interval Δt

$$\overline{V} = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} V dt$$
(7.18)

where the averaging time t is long compared to the time scale of the turbulent oscillation. The instantaneous velocity may then be written as the sum of the time-smoothed velocity \overline{V} and a velocity fluctuation V:

$$V = \overline{V} + V' \tag{7.19}$$

A similar expression can be written for the pressure, which is also fluctuating. Rewriting the equations of continuity (7.10) and momentum (7.13) for velocity and pressure by applying equation (7.19), i.e., replacing V with $\overline{V} + V'$ and P with $\overline{P} + P'$ everywhere they occur, results in the following time-smoothed equations of continuity (7.20) and motion (7.21):

$$\nabla \cdot \overline{V} = 0 \tag{7.20}$$

83

$$\rho \frac{D\overline{V}}{Dt} = \rho g - \nabla \overline{P} + \nabla \cdot \overline{\tau} \left[-\nabla \cdot \overline{\tau} \right] + R + T$$
(7.21)

The above averaged equations are almost identical in form to their instantaneous counterparts (equations (7.10) and (7.13) with the exception of the new term in equation (7.22) (in brackets). It is called the Reynolds stress tensor and characterizes the effect of turbulent eddy behavior on the mean flow. These unknown correlations appear as a result of the non-linear advection terms in the momentum equation and manifest, as noted above, the mean effect of turbulent mixing of momentum.

Equations (7.20) and (7.21) can be solved for the mean values of velocity and pressure only when the turbulence correlations can be determined in some way. In fact, the determination of these correlations is the main problem in calculating turbulent flows. A turbulence model must be introduced which approximates these correlations in some manner, typically by expressing them in terms of mean-flow quantities. Such a turbulence model together with the mean flow equations (7.20) and (7.21) form a closed set of equations for the mean values of velocity and pressure.

Many such turbulence models have been proposed and the most widely used approach to modeling the Reynolds stresses is due to Boussinesq; his eddy-viscosity concept assumes that, in analogy to the viscous stresses in laminar flow, the components of the Reynolds stress tensor are proportional to the mean velocity gradient, i.e.,

$$\overline{V}_{yx}^{(t)} = -\mu^{(t)} \frac{dV_x}{dy}$$
(7.22)

The proportionality parameter $\mu^{(t)}$ is termed the eddy viscosity and unlike the molecular viscosity μ which is a fluid property, depends on the turbulence of the flow and hence is a function of position. The eddy-viscosity concept shifts the problem of turbulence modeling to the determination of the distribution of $\mu^{(t)}$; the additional unknowns are limited to the single variable $\mu^{(t)}$.

The two-equation model incorporated into FLOTRAN is the two-equation model which is based on the Reynolds averaged equations and the eddy-viscosity concept. It is described next.

7.10 Two-equation Spalding-Launder model

In the turbulence model the turbulence field is characterized in terms of two variables, the turbulent kinetic energy k and the viscous dissipation rate of turbulent kinetic energy which are related by the Kolmogorov-Prandtl expression:

$$\mu^{(t)} = \rho c_{\mu} \frac{k^2}{\varepsilon} \tag{7.23}$$

Which relates $\mu^{(t)}$ directly to the turbulent quantities k and in the equation (7.23) above, where $c_{\mu} = 0.09$ is an empirical constant. k and following semi-empirical transport equations:

$$\rho\left(\frac{\partial k}{\partial t} + V \cdot \nabla k\right) = \nabla \cdot \left(\mu 1 + \frac{\mu^{(i)}}{\sigma_k} \nabla k\right) + \mu^{(i)} \Phi - \rho \varepsilon$$
(7.24)

$$\rho\left(\frac{\partial\varepsilon}{\partial t} + V \cdot \nabla\varepsilon\right) = \nabla \cdot \left(\mu 1 + \frac{\mu^{(t)}}{\sigma_{\varepsilon}} \nabla k\right) + c_1 \frac{\varepsilon}{k} \mu^{(t)} \Phi - \rho c_2 \frac{\varepsilon^2}{k}$$
(7.25)

where $\Phi = \nabla V$, ∇V is the rate of viscous dissipation of the mean flow and I is the unit spatial dyadic. Over the years, the model has been tested, optimized and fine tuned against a wide range of flows of practical interest. For isothermal flows with no mass transfer, this has led to the following recommended set of model constants:

$c_{\mu}=0.09$, $\sigma_{k}=1.00$, $\sigma_{\epsilon}=1.30$, $c_{1}=1.44$, $c_{2}=1.92$

which are used in FLOTRAN as the default values. It must be emphasized that these model constants are optimized for the adequate prediction of a wide range of flows. For a given flow problem, one may fine-tune these constants to obtain better agreement between model predictions and measurements. In general, however, the practice of fine-tuning is strongly discouraged. It should only be done by the expert user, who is fully aware of its potential dangers. The performance of the model is known to be very sensitive to small changes in some of the model constants (c_1 and c_2 in particular). Hence, while fine-tuning may lead to better results for a given problem, it will in general seriously degrade the universal prediction capabilities of the model and could lead to inferior predictions when small changes are made to the flow geometry and/or boundary conditions.

7.11 Boundary Conditions

When simulating turbulent flows using the model, it is particularly challenging to model the viscosity affected near-wall regions (i.e., regions adjacent to solid boundaries which contain the viscous sub-layer). A major reason is that in order to resolve the sharply varying flow variables in near-wall regions, a disproportionately large number of grid points would be required in the immediate vicinity of the solid boundary. For most typical flow scenarios this leads to prohibitively expensive computations. A second difficulty is of a primarily fundamental nature and is directly related to the type of turbulence model employed to model the effects of viscosity on the turbulence field in the viscous sub-layer (the so-called low Reynolds number effects on turbulence). The standard model, which is employed in FLOTRAN is of the high Reynolds number type and therefore cannot be used in the near-wall regions.

FLOTRAN uses a near-wall modeling methodology, which combines the accuracy and universality of the more sophisticated schemes with the cost-effectiveness of the simpler law-of-the-wall approach. In this scheme, which is based on the use of specialized elements, the computational domain is extended to the physical boundary and the full set of elliptic mean flow equations is solved all the way down to the wall. A one-elementthick layer of special elements is then employed in the near-wall region between the fully turbulent outer flow field and the physical boundary. In these special near-wall elements, specialized shape functions are used to accurately capture the sharp variations of the mean flow variables (velocities, temperature and species concentrations) in the viscosityaffected near-wall region. These specialized shape functions, which are based on the universal near-wall profiles, are functions of the characteristic turbulence Reynolds numbers and adjust automatically during the course of the computations to accurately resolve the local flow profiles. Since use is still made of the standard high Reynolds number turbulence model, the k and equations are not solved in the layer of special nearwall elements; instead, the variation of the turbulent diffusivity of momentum is modeled using Van Driest's mixing length approach.

7.12 Finite element method and equations

Only a short description of the Finite Element Method and its role in FLOTRAN can be given here due to the complexity of the subject. For a more detailed discussion of the finite element method refer to [26, 28, 30,31,32]. The following short description is adapted from [28,31,32].

The aim of the finite element method is to reduce the continuum problem (infinite number of degrees of freedom) to a discrete problem (finite number of degrees of freedom) described by a system of algebraic equations. It begins with the division of the continuum region of interest into a number of simply shaped regions called elements. Within each element, the dependent variables velocity (u_i) and pressure (p) are interpolated by functions of compatible order, in terms of values to be determined at a set of nodal points.

Within each element, the velocity and pressure are approximated by

$$\boldsymbol{u}_i(\boldsymbol{x},t) = \boldsymbol{\varphi} \boldsymbol{U}_i(t) \tag{7.26}$$

$$p(\mathbf{x},t) = \psi P(t) \tag{7.27}$$

where U_i and P are column vectors of element nodal point unknowns and ϕ and ψ are column vectors of the interpolation functions. Herein the same basis functions are

employed for all components of the velocity. Substitution of these approximations into the field equations and boundary conditions gives a set of equations:

$$f_1(\varphi, \psi, U_i, P) = R_1 \tag{7.28}$$

$$f_2(\varphi, U_i) = R_2 \tag{7.29}$$

where R_1 and R_2 are the residuals (errors) resulting from the use of the approximations of equations (7.26) and (7.27).

In FLOTRAN, the Galerkin form of the method of weighted residuals is used which seeks to reduce these errors to zero, in a weighted sense, by making the residuals orthogonal to the interpolation functions (ϕ,ψ) of each element. These orthogonality conditions are expressed by

$$(f_1, \varphi) = (R_1, \varphi) = 0$$
 (7.28)

$$(f_2, \psi) = (R_2, \psi) = 0$$
 (7.29)

where (a,b) denotes the inner product, defined by

$$(a,b) = \int_{V} a \cdot b dV \tag{7.30}$$

and V being the volume of the element.

The results of those computations can be expressed by the following matrix equations (see [61] for details):

Momentum:
$$M\dot{U} + A(U)U + K(T,U)U - CP + B(T)T = F(T)$$
 (7.31)

Mass conservation:
$$C^{T}U = 0$$
 (7.32)

where M represents the mass, A the advection (convection) of momentum, K the diffusion of momentum (K is a constant matrix for Newtonian flow and temperature (T) independence as in this study), B a buoyancy term (neglected in this work, i.e., constant), F the forcing functions for the system in terms of volume forces, respectively (see [26,31,32] for further details).

The above derivation has focused on a single finite element and the limited portion of the continuum it represents. The discrete representation of the entire continuum region of interest is obtained through an assemblage of elements such that inter-element continuity of velocity is enforced. This continuity requirement is met through the appropriate summation of equations for nodes common to adjacent elements - the direct stiffness approach. The result of such an assembly process is a system of matrix equations.

7.13 3D elements

The tetrahedron elements available in the computational scheme (FLOTRAN) include a 4 node linear element as shown in figure 7.2. The interpolation functions are expressed in terms of the natural coordinates (L₁, L₂, L₃, L₄); the natural coordinates L_i are not independent but related by $L_1 + L_2 + L_3 + L_4 = 1$.



Figure 7.2 Four noded tetrahedron element

The velocity approximation used with this element is continuous. A linear interpolation, ϕ , of the linear is employed with the velocity degrees of freedom being located at the corner vertices of the tetrahedron. The velocity interpolation functions are those of equation are given by.

$$\varphi = \begin{pmatrix} L_1 \\ L_2 \\ L_3 \\ L_4 \end{pmatrix}$$
(7.33)

Similarly the pressure interpolation functions are given by.

$$\psi = \begin{pmatrix} L_1 \\ L_2 \\ L_3 \\ L_4 \end{pmatrix}$$
(7.34)

The iso-parametric concept is used to define the coordinate transformations,

$$x=Nx, y=Ny, z=Nz; N=N(L_i)$$
(7.35)

In the computational scheme FLOTRAN, the 4 node tetrahedrons are all iso-parametric elements; i.e., $N = \phi = \psi$.

The relationship between derivatives with respect to the physical coordinates (x,y,z) and derivatives with respect to the normalized

coordinates (r,s,t) is defined by

$$\begin{pmatrix} \frac{\partial \varphi}{\partial r} \\ \frac{\partial \varphi}{\partial s} \\ \frac{\partial \varphi}{\partial s} \\ \frac{\partial \varphi}{\partial t} \end{pmatrix} = \begin{pmatrix} \frac{\partial x}{\partial r} & \frac{\partial y}{\partial r} & \frac{\partial z}{\partial r} \\ \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s} & \frac{\partial z}{\partial s} \\ \frac{\partial \varphi}{\partial t} & \frac{\partial \varphi}{\partial t} & \frac{\partial \varphi}{\partial t} \end{pmatrix} = \begin{pmatrix} \frac{\partial N^{T}}{\partial r} X & \frac{\partial N^{T}}{\partial r} Y & \frac{\partial N^{T}}{\partial r} Z \\ \frac{\partial N^{T}}{\partial s} X & \frac{\partial N^{T}}{\partial s} Y & \frac{\partial N^{T}}{\partial s} Z \\ \frac{\partial N^{T}}{\partial t} X & \frac{\partial N^{T}}{\partial t} Y & \frac{\partial N^{T}}{\partial t} Z \end{pmatrix} \begin{pmatrix} \frac{\partial \varphi}{\partial x} \\ \frac{\partial \varphi}{\partial y} \\ \frac{\partial \varphi}{\partial y} \\ \frac{\partial \varphi}{\partial z} \end{pmatrix} = J \begin{pmatrix} \frac{\partial \varphi}{\partial x} \\ \frac{\partial \varphi}{\partial y} \\ \frac{\partial \varphi}{\partial y} \\ \frac{\partial \varphi}{\partial z} \end{pmatrix}$$
(7.36)

where J is the Jacobian matrix. Inverting the Jacobian matrix provides the required relation for the derivatives of the basis functions,

$$\begin{pmatrix} \frac{\partial \varphi}{\partial x} \\ \frac{\partial \varphi}{\partial y} \\ \frac{\partial \varphi}{\partial z} \end{pmatrix} = J^{-1} \begin{pmatrix} \frac{\partial \varphi}{\partial r} \\ \frac{\partial \varphi}{\partial s} \\ \frac{\partial \varphi}{\partial t} \end{pmatrix}$$
(7.37)

For integral evaluation the elemental volume is required and given by

$$dxdydz = |J|drdsdt$$
; $|J| = Determinant of J$ (7.38)

7.14 Shape functions

FLOTRAN approximates the correct velocities and pressures at all of the nodes so that the mass, momentum and energy equations are satisfied. However, this is only for the nodes, it calculates the values for the velocities and presures for all the points within the element using shape functions. There is a shape function for each node in a particular element . The shape functions are derived such that they have a value of unity at the node they are associated with, and a value of unity to zero at all other nodes in the element, they may be allowed to vary from unity to zero in a linear or higher order fashion. Therefore the value of velocity or pressure may be obtained by calculating the value of each shape function at the point in question, and the multiplying it by the velocity or pressure at it's associated node. When we sum up the contribution from each node we obtain an accurate picture for the complete distribution of the variable in question. So in summary, shape functions enable us to convert a discrete set of values (at nodes) in to a smooth continuum like distribution across each element.

7.15 Discretization of equations

The momentum equations, energy, and turbulence equations have the form of a scalar transport equation.

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho V \phi) - \nabla \cdot (D(x, y, z) \nabla \phi) = S$$
(7.39)

This equation represents a number of fluid flow equations

• Degree of freedom	Equation	D	S
1	Mass conserv.		0
u	x-momentum	μ	$\rho g_x - \frac{\partial p}{\partial x} + R_x$
V	y-momentum	μ	$\rho g_y - \frac{\partial p}{\partial y} + R_y$
w	z-momentum	μ	$\rho g_z - \frac{\partial p}{\partial z} + R_z$
Т	Temp.(energy)	K	q
k	Turb. K.E	$rac{\mu^{(t)}}{\sigma_k}$	$\mu^{(t)}G - \rho \varepsilon$
ε	T.KE. Diss. Rate	$rac{\mu^{(t)}}{\sigma_{arepsilon}}$	$c_1 \mu^{(t)} \frac{\varepsilon}{k} G - \frac{c_2 \rho \varepsilon^2}{k}$

Where:

k = Turbulent kinetic energy

K = Thermal conductivty

$$G = \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \frac{\partial u_i}{\partial x_j}$$
(7.40)

 $\mu^{(t)}$ = Turbulent viscosity

 σ_{ε} , σ_k , C_1 , and C_2 are coefficients from the Spalding and Launder κ - ε turbulence model.

7.16 FLOTRAN analysis

A typical FLOTRAN analysis consists of several steps. To illustrate this process the steps involved in performing an analysis of 2D steady-state turbulent flow of water around a NACA0021 hydrofoil disposed at an angle of attack of -10° are shown in figures 7.3-7.12

(i)Setup

Define the element type. FLOTRAN supports the FLUID141 element (4-noded quadrilateral and/or 3-noded triangular) elements for two-dimensional models, and FLUID142 element (8-noded hexahedral and/or 4-noded tetrahedral) elements for three-dimensional models. For three-dimensional analyses, only all hexahedral or tetrahedral meshes are allowed. The quadrilateral and triangular elements may be mixed in the two dimensional analyses. The degrees of freedom of elements FLUID141 and FLUID142 include fluid velocity, pressure, temperature, turbulent kinetic energy, turbulent kinetic dissipation and multiple species mass fractions for up to six fluids.

(ii) Define geometry; generate the nodes and elements.

The finite element method requires that the physical space/domain be slip into small volumes, the ``elements". Finite element methods generally use irregular meshes: there is no special advantage in using regular meshes.

Mesh characteristics

For all types of meshes, control of certain characteristics is wanted:

• *The local density of points*. High density gives more accuracy, but computation takes longer.

- The smoothness of the point distribution. Large variations in grid density or shape can cause numerical diffusion or anti-diffusion and dispersion or refraction of waves. This can lead to inaccurate results or instability.
- *The shape of the grid volumes.* For instance, boundary layers in fluid flow require a grid that is very compressed normal to the flow direction. In the finite element method using triangular elements, the maximum angle must be bounded strictly below to prove convergence of the method as the element size is reduced.
- *For simple domains*. The choice between regular or irregular meshes is governed mainly by the discretization method. However, for complex domains (as present in this thesis) irregular meshes are preferred to regular meshes (as they are also much easier to generate automatically). Irregular mesh generation (at least for triangular or tetrahedral elements) can be fully automatic and fast. Regular mesh generation requires the domain to be split up into simple blocks (multi-block) which are then meshed automatically.

Structured meshes

Structured meshes are characterized by regular connectivity, i.e., the points of the grid can be indexed (by 2 indices in 2D, 3 indices in 3D) and the neighbors of each point can be calculated rather than looked up (i.e., the neighbors of the point (i,j) are at (i+1,j), (i-1,j), etc.). The regularity of structured grids allows fast solvers to be used. There is a large user effort in constructing the multi-block decomposition and the tendency for complex configurations is now to use unstructured meshes, which can be automatically generated, even though the solvers are slower.

Unstructured meshes

Unstructured meshes have been developed mainly for the finite element method. There is a large range of possible shapes for finite elements: tetrahedra, prisms, blocks, and there can be arbitrary connectivity, leading to unstructured meshes. However, the only shapes that can be used to generate meshes fully automatically are triangles in 2D and tetrahedra in 3D. Although block elements are desirable, it is much harder to generate grids of block elements automatically. Therefore only triangular/tetrahedral mesh generation is further considered.

Mesh requirements for the Finite Element Method

The Finite Element Method (FEM) has certain requirements on a mesh: The mesh must be valid, i.e., no holes, no self-intersection, no faces joined at two or more edges, etc. This is an obvious requirement, but many mesh generation schemes require (and their implementations contain) a large amount of checking for these conditions.

The mesh must conform to the boundary of the domain.

The density of the mesh must be controllable, to allow trade-off between accuracy and solution time.

The mesh density will vary depending on local accuracy requirements, but this variation must be smooth to reduce or eliminate numerical diffusion/refraction effects.

There are some requirements on the shape of elements. In general, the elements should as "equiangular" as possible, e.g., equilateral triangles, regular tetrahedra. Highly distorted elements, e.g., long, thin triangles, squashed tetrahedra, can lead to numerical stability problems caused by round-off errors. These requirements are modified for boundary layers, where highly stretched elements are desired and the FEM formulation allows for them. Even in this case, the min-max-angle property is required. Triangles/tetrahedra can fit irregular boundaries and allow a progressive change of element size without excessive distortion, and are therefore well-suited for mesh generation for FEM. In addition, there are fully-automatic methods for generating triangular/tetrahedral meshes. However, linear tetrahedra are not that good for FEM (they are too "stiff") and a high density of elements is needed to give acceptable results; this leads to increased solver time. Hexahedral elements are much better, but it is difficult to automatically generate all-hexahedral meshes (even allowing transition elements such as prisms).







Figure 7.3 Divide the domain into regions where you might expect high or low gradients in the solution variables.



Figure 7.4 An O-type structured mesh, this mesh has a regular connectivity, i.e., each point has the same number of neighbors (for some grids a small number of points will have a different number of neighbors).



Figure 7.5 An unstructured mesh, this mesh a regular connectivity, i.e., each point has the same number of neighbors (for some grids a small number of points will have a different number of neighbors).

(iii) Apply boundary conditions

Velocities (X, Y, and Z direction), pressure, temperature, turbulent kinetic energy, and turbulent kinetic dissipation rate are degrees of freedom constraints specified at points where their values are known. Velocities or pressures are usually specified at the flow inlet or outlet boundaries. The presence of non slip boundaries are indicated by specifying zero velocities at the nodes representing the wall.



Figure 7.6 Inlet boundary conditions applied to the front 180° , (i.e. $V_x = 10$, $V_y = 0$), and the outlet boundary conditions are applied to rear to the rear 180° , (i.e. Pressure = 0).



Figure 7.7 Non slip boundary conditions are applied to the surface of the hydrofoil (i.e. $V_x = 0, V_y = 0$).

When the model is completed and all the boundary conditions are defined the fluid properties and flow analysis parameters must be specified. Fluid properties may be constant or variable, properties which need to be defined include density, viscosity, conductivity, and so on. The flow analysis parameters include operating conditions (steady or unsteady flow, reference pressure, acceleration, ambient temperature, etc.), turbulence values, relaxation parameters, solution (specify solvers, no. of global, quadrature order iterations, convergence criteria, stop time, etc).Numerical methods used to solve the equations for fluid flow iteration procedures. By their nature, iterative solution methods require convergence criteria that are used to decide when the iterations can be terminated.

In many cases, iteration methods are supplemented with relaxation techniques. For example, over relaxation is often used to accelerate the convergence of pressure-velocity iteration methods, which are needed to satisfy an incompressible flow condition. Under relaxation is sometimes used to achieve numerically stable results when all the flow equations are implicitly coupled together. The amount of over or under relaxation used can be critical. Too much, leads to numerical instabilities, while too little slows down convergence. Similarly, a poorly chosen convergence criteria can lead to either poor results (when too loose) or excessive computational times (when too tight). Selecting proper relaxation and convergence criteria can be a difficult and frustrating experience. The criteria depend on the specifics of the problem being solved, and they may change during the evolution of a problem. Unfortunately, there are no universal guidelines for selecting criteria because they depend not only on the physical processes being approximated, but also on the details of the numerical formulation. FLOTRAN has a standard set of recommended criteria, but users must often resort to trial-and-error adjustments to get good results.

(iv) Solving the problem

You can monitor solution and stability of the analysis by observing the rate of change of the solution and the behavior of the relevant dependent variables include these velocities (X, Y, and Z direction), pressure, temperature, turbulent kinetic energy, and turbulent kinetic dissipation rate. FLOTRAN calculates convergence monitors for each degree of freedom, for every global iteration. The convergence monitors are a normalised measure of the solutions rate of change from one iteration to the next. Denoting by the field variable, Φ , as any degree of freedom, the convergence monitor is defined as follows:

Convergence monitor =
$$\sum_{i=1}^{n} \frac{\left| \Phi_{i}^{k} - \Phi_{i}^{k-1} \right|}{\left| \Phi_{i}^{k} \right|}$$
(7.41)

Therefore convergence monitor represents the sum of the of changes of the variable calculated from the results between the current k^{th} and the $(k-1)^{th}$ iteration, divided by the sum of the current values. This is preformed over all n nodes, using the absolute values of the differences. This convergence monitor information displayed on a semi-log style graph. Figures show two typical examples figure 7.8 (a) shows a rapidly converging 2D steady state laminar (only three convergence monitors are displayed in figure 7.8 (b), since there are only three field variables, i.e. V_x , V_y , and pressure). While figure 7.8(b) shows a very slowly converging 3D steady –state turbulent simulation,



Figure 7.8 convergence monitor information displays



Figure 7.9 Iterative algorithm solution.

(v) Reviewing results

Once a converged solution has been obtained the results may be viewed using ANSYS'S postprocessor **/post1**, which capable of displaying the completed flow simulation in several ways. They include velocity vectors, streamlines, line or filled contours, profiles, iso-surfaces, and x-y plots. Examples of some of these are shown in figures 7.10-12.



Figures 7.10 Velocity vectors at trailing edge ofydrofoil.



Figures 7.11 Streamlines around hydrofoil.



Figures 7.12 Contours showing the spatial pressure distribution about the hydrofoil.

7.17 Three-dimensional CFD analysis of the U-Kite.

The ultimate objective of the F.E. analysis of the flow surrounding the U-Kite is determine the lift, drag, and side-slip/force generated by its motion through the water. The overall strategy used for this is to

• Calculate nodal velocities
- Calculate nodal pressures
- Integrate the nodal pressures at the surface of the U-Kite, to determine lift, drag, and side-slip.

When modeling the flow around the U-Kite using FLOTRAN the following were assumed

- (i) The fluid was incompressible
- (ii) Steady state conditions dominate the flow around the U-Kite.
- (iii) Acceleration due gravity was 9.81 m/s
- (iv) The ambient temperature of the fluid surrounding U-Kite was 20°C.
- (v) The density of the fluid was 1.025 kg/m^3 .
- (vii) The viscosity of the fluid was 1.07×10^{-3} kg/ms.

When faced with a complex problem, the natural tendency is to try to include, all aspects of the analysis immediately. However, my experience has shown me that if I try to include too many features at once, I may not have a solid basis for my solutions and/or I may not be able to achieve a solution in a timely manner. A better approach is to incrementally add features to a small model until I am confident in solutions derived. Once I have established a valid solution, with a small model, then it is time to try a larger model.

7.18 Geometry definition, and Model prepartion.

Taking advantage of the fact that ANSYS/FLOTRAN can accept the IGES standard files, which Pro/Engineer is capable of creating. It was possible to import the line, point and surface entities, which go to make up the external surfaces of the U-kite into ANSYS/FLOTRAN. Using these surfaces it was possible to then construct a suitable fluid domain using these imported surfaces as part of the boundary. It is imperative to understand that it is the fluid region surrounding the U-Kite that is being modeled and not the U-kite itself. The U-Kite surfaces serve only as a boundary to the flow. The computational domain or fluid region must be large enough to approximate a free-stream environment yet small enough to minimize computation time.



Figure 7.13 Fluid domain.

For simplicity, a rectangular block was selected as the fluid region boundary as shown in Figure 7.13. It was divided into a number of smaller volumes or fluid regions; this was done to make the task of meshing easier. Once the geometry had been completely defined the task of mesh generation was undertaken. Mesh density was controlled by specifying the number, and the spacing of elements along lines which made up the individual boundaries of the domain, so that the node density increased in areas where high fluid dynamic gradients are anticipated e.g. corners, and surfaces. Parameters controlling the aspect and transition ratio and distortion of elements were also entered. The domain was meshed using 80,000 four noded tetrahedral elements (FLUID 142), this element allows

for fast, accurate representation of the domain boundaries, and yield results of sufficient accuracy. The above meshed model was checked for coincident nodes, coincident elements, and element distortion. The actual process of meshing took more than three weeks before a satisfactory mesh with the desired distribution and mesh ratio (i.e. where elements gradually become larger) of elements was finally obtained, the division, and spacing parameters had to be changed several times.



Figure 7.14 Meshed domain.



Figure 7.15 Element faces at U-Kite surface

To approximate the free-stream environment, free-stream boundary conditions were enforced on five faces of the rectangular block, including the upstream face, these conditions were applied as V_x = free stream velocity, and $V_y = V_z = 0$. The non-slip and no-penetration conditions were applied as V_x = $V_y = V_z = 0$ on the surfaces of the U-Kite. Finally on the downstream face the pressure was set to zero.

Having completed applying the boundary conditions the relevant fluid properties, flow parameters, and convergence criteria were entered. Because this is a high Reynolds number flow, the problem was pre-conditioned to prevent divergence of the solution. Preconditioning was accomplished by first obtaining an approximate laminar solution (Re between 100-1000 and then gradually introducing the turbulence. The practical method for setting up the laminar problem is to leave all values unchanged except viscosity, which is raised by the appropriate order of magnitude. The intent is to raise the level of the viscosity to a level, which will produce a stable, laminar result. Generally an increase of a few orders of magnitude will be sufficient Usually an approximate laminar solution can be achieved in 10 -20 global iterations. After an approximate laminar solution is achieved, the viscosity, should be decreased by one order of magnitude, and the problem run again. This process should be continued until a Reynolds number of 1000-2000 is achieved. The convergence parameters should be monitored throughout the process to ensure that the solution has remained stable and is converging. It is helpful to under relax the solution when changing these properties. Now that an approximate laminar solution has been achieved, turbulence effects can now be slowly added, and the viscosity can be adjusted as specified above with all runs restarting from the previous set of results to establish a reasonable k-E field. This process is continued until the viscosity has reached the real value. It is not necessary to achieve global convergence for the intermediate viscosity runs adjusted.

7.19 Results

Analyses were conducted at towing speeds between 0.5 and 8 m/s, beyond 8 m/s the solutions failed to meet the convergence criteria. The results are summarized in the two graphs below the first graph shows the variation of lift, drag, and side-slip with towing velocity for a U-Kite whose lower and upper foils were disposed at angles of -2° and -4° respectively to the direction of the water flow. While the second graph shows the variation of lift, drag, and side-slip with towing velocity for a U-Kite whose lower and upper foils were disposed at angles of -2° and -4° respectively to the direction of the water flow. While the second graph shows the variation of lift, drag, and side-slip with towing velocity for a U-Kite whose lower and upper foils were both disposed at an angle of -2° to the direction of the flow.



Figure 7.16 variation of hydrodynamic forces with towing velocity for a U-Kite whose lower and upper foils were disposed at angles of attack of -2° and -4° respectively.



Figure 7.17 variation of hydrodynamic forces with towing velocity for a U-Kite whose lower and upper foils were both disposed at an angle of attack of -2° .

These graphs show that

- (i) There is a non-linear relationship between towing speed and the hydrodynamic forces.
- (ii) Doubling the towing speed will result in an approximately four-fold increase in hydrodynamic forces acting on the U-Kite.
- (iii) The decrease of the angle of attack of the upper appears to have little effect on drag.
- (iv) Changing the angle attack of the upper wing from -4° to -2° results in the lift generated by U-Kite being nearly halved.

Figure 7.18 shows the variation of pressure over the U-Kite's surface. The maximum value of pressure (indicated by bright red) occurs on the hemispherical nose of the vehicle, other areas where high pressures arise are at the fronts of the buoyancy and ballast pods. Reducing the frontal areas of these (streamlining) would lead to a reduction in drag. Low values of pressure occur on the underside of the foils (indicated by blue) as expected. Minimum pressure values occur at the junction between the foils and the pods. This would seem to indicate that the pods at the ends of the foils have reduced or eliminated the formation and shedding of foil tip vortices, and have effectively increased aspect ratio of the foils. A close examination of figure 7.18 reveals that the lowest values tend to occur along or close to the quarter chord line of the hydrofoil (on aircraft this is referred to as the aerodynamic centre-line).

While it is not possible to validate these results without a tow tank/wind tunnel, the magnitude of the lift generated by individual foils is very close to those calculated using lifting line theory (see section 4.5 and Appendix D).



Figure 7.18 Pressure distribution over U-Kite surface, towing velocity 0.5 m/s, angle of attack of upper hydrofoil -4°, lower hydrofoil -2°.

7.18 Design of upper hydrofoil

Using the finite element results it was possible to examine the pressures, the resultant forces, and their components acting on individual parts which make up the vehicle, since the mesh has being constructed so that the software can distinguish between them. The graph below shows the variation of the forces acting on the upper hydrofoil with towing

velocity.



Figure 7.19 Surface pressure distribution for the upper hydrofoil.





The diagram in figure 7.21, illustrates how the longitudinal structural members inside the hydrofoil support the loads encountered, and the means by which these loads are transmitted from the skins to structural members. Clearly the total effective cross-sectional area of these longitudinal members must be large enough to resist bending moments, and axial stresses. The method employed for determining the minimum cross-sectional area is given below



Figure 7.21 Longitudinal structural members inside the hydrofoil

The longitudinal members may be modeled as a single cantilever with a uniform circular cross section, with the hydrodynamic load acting at a point 200mm from secured end. This assumption is justified by the fact that the maximum deflection will occur at the end of the structural member, and consequently the maximum bending stress will occur at opposite end of the member.



Figure 7.22 Simplified representation of the load acting on the structural members

When the U-Kite is traveling at 5m/s (i.e. the maximum design towing speed), the magnitude of the resultant hydrodynamic load acting on the upper hydrofoil whose angle of attack is -4°, is typically 300 N.

Using simple bending theory

$$\frac{2\sigma}{Dia.} = \frac{M}{I} \tag{7.42}$$

112

$$\frac{2\sigma}{Dia.} = \frac{32M}{\pi(Dia.^4)} \tag{7.43}$$

Where

$$\sigma = \frac{\sigma_y}{F.O.S} \tag{7.44}$$

Assuming the member is made of aluminium alloy 5052 H38,

 σ_y = Yield stress = 248 N/mm² F.O.S = Factor of safety = 3 $\Rightarrow \sigma$ = Maximum stress \approx 83 N/mm² M = Bending moment = 300N × 200 mm = 60,000 Nmm Dia = Rod diameter = 17.95 mm

So long as one of the members inside the hydrofoil has a diameter of 18mm, the hydrofoils should be more than capable of supporting any hydrodynamic load encountered during normal use.

Chapter 8

Conclusions

8.1 Field trial

The prototype U-Kite system was subjected to a limited field trial in september, 1998 In order to evaluate the mechanical design, the stability of the towed body and the performance of the hydrofoils. These trials were conducted in Galway Bay half a mile south of Galway city. The system was towed along side a rigid inflatible boat to test stability and buoyancy prior to the installation of any instrumentation. Once the vehicle was deemed seaworthy, a pressure transducer was mounted inside the body(where the pressure is equal to the static pressure, which is directly proportional to the column of water above).

Fully operational sea trials commenced after making several adjustments to the position at which the towing line was attached to the bridle, and removing ballast to reduce weight, and to ensure the system was neutrally buoyant, and the vehicle floated in the horizontal position when placed in sea-water. The angle of attack of the upper hydrofoil was set to -4° , and the lower foil was set to -2° .

Using a manually operated hand winch, 25 metres of unfaired cable was paid out. Within a short time after deploying the U-Kite, it showed an operational performance beyond most expectations. The U-Kite was towed at speeds between 3 and 4.5 meters per second, and water depths varied from 10 to 13 meters. During these tests, the depth of the vehicle could be controlled by varying the length of the amount of cable paid out.

As mentioned above these trials were limited, the lack of suitable instrumentation such as accelerometers prevented the measurement of the pitch roll and yaw of the towed body during the trials so the stability of the U-Kite could be assessed by observing the behaviour of the U-Kite when close to the surface. Further trials need to be conducted to asses the vehicles stability.

8.2 Conclusions

The primary goal of this project was to develop a working system, which is capable of carrying oceanographic sensors. Presently work on the U-Kite is still ongoing, and it is

hoped that further field trials at sea will commence in the very near future. A prototype was built using CADCAM technology, and strong composite materials, and subsequently launched at the Oceanology international conference in Brighton April '98, where it created a lot of interest amongst oceanography community.

During the course of this project, many different aspects of computer-aided engineering were used in the design of a towed submersible oceanographic instrument platform. These aspects included three-dimensional solid modeling, Computational Fluid Dynamics, and finally computer integrated manufacturing. All 3-D models or parts, which make up the U-Kite, had been developed using the parametric features based solid modeler Pro/Engineer, release 18. The detailed parts were then assembled in the assembly section of the package, and clearances between parts were verified. All 2-D drawings of the components and assembly were made using Pro/Engineer's drawing module.

Pro/Engineer allows the designer to conveniently create, modify, store, and retrieve very complex from the appropriate database. The ability to visualize a fully defined solid model representation of assembled components would ensure that design errors can be avoided which may prove to be disastrous during the assembly stage of manufacture.

Chapter 7 demonstrates the utilization of three-dimensional computational dynamics to provide a clear understanding of the operation of the U-Kite, and in the design of critical components, namely the loading bearing structures inside the hydrofoils of the towed vehicle. During the course of these CFD investigations, the many barriers that limit to the use of CFD in design were encountered. These included: complex model building is labour- intensive and slow, computational turnaround times can be long, and passing data to/from CAD and other CAE packages can be cumbersome, slow solution convergence, and solution divergence. Despite these disadvantages, CFD is a viable engineering tool as it eliminates the need to build several physical prototypes, since implementing and testing design modifications in software takes a fraction of the time and cost.

Pro/Engineer surface models were fed into ANSYS/FLOTRAN, which allowed me to study water-flow at various speeds and using slightly different hydrofoil configurations. The results were loaded into ANSYS'S postprocessor, where it was possible to generate spatial cuts on the vehicle's surface to show surface pressure at different locations. The computer graphics of the results clearly illustrated the forces at work on the vehicle. These graphics were a strong tool for understanding the complex issues associated with travelling underwater.

While it is not possible to validate these results from the CFD analyses without a tow tank/wind tunnel, the magnitude of the lift generated by individual foils of the U-Kite were not too dissimilar to those obtained using line theory (see section 4.5 and appendix D).

The multi-step procedure using a combination of modern and traditional methods for making fibre-reinforced plastic hydrofoils described in chapters 5, and 6, was also used to fabricate components such as nose sections, tail cones for the vehicle. There are other viable methods available, but considering that CNC machines are ever more accessible, and that CAD/CAM software is becoming more competitively priced. This method has definite practical advantages for producing quality components cheaply in small quantities.

The U-Kite was designed to meet a number of specific requirements, many of which were specified to permit survey operations in shallow estuarine waters. Following a series of field trials in Galway, these trials proved to be very successful. The vehicle's mechanical design permitted disassembly, transportation and re-assembly on a small open boat without any significant problems. Vehicle launch and recovery procedures went better than planned. It was successfully demonstrated that the U-Kite could be used in underwater operations.

The U-Kite prototype is a towed body that can meet many commercial and environmental survey applications in its current state. U-Kite started from the identification of a need, in both commercial and environmental applications, for a cost-effective, stable, adaptable towed submersible instrument platform.

With only eighteen months to complete the project, I was not afforded the luxury of design iteration and testing. It is my contention that the success of this project was a direct result of the persistent use of rational engineering.

References

- 1) Marine technology reference book, N. Morgan, Butterworths, 1990.
- Handbook of ocean and underwater engineering, J. J. Myers et al, Mc Graw-Hill, 1969.
- 3) Underwater work systems, H. R. Talkington, Marcel Dekker, 1981.
- N-v Shuttle Enhanced undulating vehicle, Sea Technology, J. Dunning, Vol. 39 No. 2, 1998
- Three-dimensional analysis of towed system, Journal of Hydronautics, Vol.2 part
 4. 1968.
- Dynamic applications of towed vehicles, engineered solutions for varied tasks, 3 case studies, Underwater association of diving contractors magazine, W. H. Doyle III, Vol. X No. 1 1998.
- Motion of underwater towed system, N. Kato, Proceedings of the 5th international offshore mechanics and Arctic engineering symposium. Vol. III ASME.
- 8) Thinking Pro/Engineer, mastering design methodology, Onward Press 1995.
- Pro/Engineering Part modeling users guide, release 18, Parametric Technology Corporation, 1995.
- Pro/Manufacture / NC-Check users guide, release 18, Parametric technology corporation, 1995.
- Pro/Engineering Drawing users guide, release 18, Parametric technology corporation, 1995.
- 12) Aerodynamics for engineers, J.J. Bertin, & M. L. Smith, Prentice-Hall Inc., 1989.
- 13) Theory of wing sections, H. I. Abbott, & A. E. Von Doenhoff, Dover, 1959.
- 14) Mathematica as a tool, S. Kaufmann, Birkhauser Verlag, 1994.
- 15) Calculus to chaos, an introduction to dynamics, D. Acheson, Oxford University press, 1997
- 16) Viscous fluid flow, F. White, Mc Graw-Hill, 1986.
- 17) Fluid-Dynamic Drag, S.F.Hoener, Brick Town, 1965
- 18) Underwater towed-vehicle Maneuverable in both the vertical and horizontal axis,
 N. Kato, 1st ISOPE Conference, vol. 2, PP 85-92.
- 19) Marine corrosion -Causes and prevention, F. L. LaQue, John Wiley, 1975.

- 20) Integrated computer-aided design of Mechanical systems, S. A. Meguid, Elsevier applied science, 1986.
- 21) NC machine programming and software design, C.H. Chang & M. Melkanoff, Prentice-Hall Inc.
- 22) Computer-aided deign and Manufacture, C. B. Besant, & C. W. Lui, Ellis Horwood, 1986.
- 23) Manufacturing of polymer composites, B. Astrom, 1997.
- 24) Light airplane construction for amateur builders, L. Pazmany, 1970.
- 25) Pro-Set laminating epoxy handling guide, Gougeon Brothers, Inc., 1996.
- 26) Finite element computational fluid mechanics, Mc Graw-Hill, 1983.
- 27) Combining fluid flow and structural analysis to provide the ultimate wind surfing experience, S. Fagg, & D. Parish, ANSYS News, fourth issue, 1996.
- 28) The finite element method for engineers, Huebener, Thorton, & Byrom, 3rd edition.
- 29) Numerical analysis, R. Burden, J. D. Faires, Brookes/Cole publishing co., 1997.
- 30) FLOTRAN Users manual, Compuflo Inc. 1992.
- 31) FLOTRAN theoretical manual, 1992.
- 32) ANSYS theory reference, seventh edition, SAS IP, Inc.
- 33) The Navier stokes equations, C.Taylor, T.G Hughes, Pine Bridge Press, 1981.

Appendix A Hydrofoil section definition

NACA 4 Airfoil Sections

The NACA 4 airfoils represent a family of airfoil sections that can be generated by the use of a set of simple polynomial equations. While these sections are slightly out of date in terms of current aircraft usage, but they are still widely used by the yacht designers, these are useful sections and are easy to create. The airfoils are created by summing a thickness distribution with a given mean line equation. For NACA 4 airfoil sections the thickness distribution is as follows,



Figure1 NACA 4 series airfoil basic thickness form.



Figure 2 NACA 0021 Basic thickness form.

Where x is a position along the chord line, given as a fraction of chord and t is the value of maximum thickness as given by the last two digits of the airfoil designation number.(ie 0012 = symmetric section with t(max)=0.12c). The mean line is given as,

$$y_{c} = (m / (p^{2})) (2 px - x^{2})$$
for 0y_{c} = (m / (1 - p)^{2}) [(1 - 2 p) + 2 px - x^{2}] for p

Values p and m are given from the first two digits of the designation number. m being the value of maximum camber height (1/100ths chord) and p being the position of maximum camber height

(1/10ths chord). (ie 2412 = maximum camber height =0.02c located at 0.4c). The following Qbasic program uses the above equations to construct the desired 4 NACA

airfoil section. The user inputs the desired section designation number and the number of surface data points required. Points are then generated using a cosine distribution of chord x coordinates. For each x coordinate an upper (xu,yu) and lower surface (xl,yl) data point is created by applying the above equations and construction method.

 $xu = x - y_t \sin (q)$ $yu = y_c + y_t \cos (q)$ $xl = x + y_t \sin (q)$ $yl = y_c - y_t \cos (q)$

A leading edge radius r is applied to smooth the front data points.

$$r_t = 1.10119 t^2$$

The coordinate data that is created then be stored as an ASCII formatted IBL file which can be directly imported into Pro/Engineer to construct a datum curve.

```
1000 PI = 4 * ATN(1)
1010 NP = 100
1020 \text{ MPTS} = 400
1030 \text{ DTP} = 3
1040 DIM XCC(MPTS), XU(MPTS), YU(MPTS), XL(MPTS), YL(MPTS), YT(MPTS), YC(MPTS)
1050 DIM XDD(MPTS), DYC(MPTS)
1060 DELTH = PI / (NPP - 1)
1070 FOR I = 1 TO NPP
1080 \text{ XDD}(I) = .5 - .5 * \text{COS}(\text{DELTH} * (I - 1))
1090 NEXT I
2000 ' REJOIN PATHS
2100 DELTH = PI / NP
2110 FOR I = 1 TO NP
2120 XCC(I) = .5 - .5 * COS(DELTH * I)
2130 NEXT I
2140 '
2150 ' NACA 4 DIGIT -----
2160 '
2170 INPUT " Enter The Chord Lenght"; CHORD
2180 INPUT " Enter The First Digit Of The 4 Digit Designation"; MM
2190 INPUT " Enter The Second Digit Of The 4 Digit Designation"; PP
2200 INPUT "Enter The Last Two Digits Of The 4 Digit Designation"; TOC
2210 PRINT ""
2220 IF MM = 0 THEN
2230 GOTO 2770
2240 END IF
2250 MC = MM / 100
2260 PC = PP / 10
2270 TC = TOC / 100
2280 FOR I = 1 TO NP
2290 YT(I) = TC * (1.4845 * SQR(XCC(I)) - .63 * XCC(I) - 1.758 * XCC(I) ^ 2 + 1.4215 * XCC(I) ^ 3 -
.5075 * XCC(I) ^ 4)
```

```
2300 IF MC = 0 THEN
2310 \text{ YC}(I) = 0
2320 \text{ DYC}(I) = 0
2330 ELSE
2340 IF XCC(I) > PC THEN
2350 YC(I) = MC / (1 - PC) ^ 2 * (1 - 2 * PC + 2 * PC * XCC(I) - XCC(I) ^ 2)
2360 DYC(I) = 2 * MC / (1 - PC) ^ 2 * (PC - XCC(I))
2370 ELSE
2380 YC(I) = MC / PC ^2 (2 * PC * XCC(I) - XCC(I) ^2)
2390 DYC(I) = 2 * MC / PC ^ 2 * (PC - XCC(I))
2400 END IF
2410 END IF
2420 NEXT I
2430 LER = 1.1019 * TC ^ 2
2440 TEANG = 2 * ATN(1.16925 * TC)
2450 DESIG = MM * 1000 + PP * 100 + TOC
2460 DESIG$ = STR$(DESIG)
2470 FOR I = 1 TO NP
2480 \text{ THET} = \text{ATN}(\text{DYC}(I))
2490 XU(I) = XCC(I) - YT(I) * SIN(THET)
2500 \text{ YU}(I) = \text{YC}(I) + \text{YT}(I) * \text{COS}(\text{THET})
2510 \text{ XL}(I) = \text{XCC}(I) + \text{YT}(I) * \text{SIN}(THET)
2520 \text{ YL}(I) = \text{YC}(I) - \text{YT}(I) * \text{COS}(\text{THET})
2530 NEXT I
2540 \text{ NPP} = \text{NP}
2550 GOTO 2560
2560 CLS : LOCATE 12, 18
2570 CLS : PRINT
2580 INPUT " Enter The Filename For Output Of The Coordinates Include .ibl prefix"; DEST$
2590 OPEN DEST$ FOR OUTPUT AS #1
2600 GOSUB 2630
2610 CLOSE #1
2620 END
2630 PRINT #1, "OPEN"
2640 PRINT #1, "ARCLENGTH"
2650 PRINT #1, "BEGINSECTION! 1"
2660 PRINT #1, "BEGINCURVE"
2670 FOR I = 1 TO NPP
2680 PRINT #1, USING "### ###.### ###.### #"; I; CHORD * XU(I); CHORD * YU(I); 0 * YU(I)
2690 NEXT I
2700 PRINT #1, ""
2710 PRINT #1, "BEGINSECTION! 2"
2720 PRINT #1, "BEGINCURVE"
2730 FOR I = 1 TO NPP
2740 PRINT #1, USING "### ###.### ###.### #"; 1; CHORD * XL(101 - I); CHORD * YL(101 - I); 0 *
YL(I)
2750 NEXT I
2760 RETURN
2770 \text{ XCC}(I) = \text{XDD}(I)
2780 GOTO 2270
```

iv

Appendix B U-Kite Drawings



	HR.
	and the second s
	\bigcirc
DESIGNER 28-Ma-99 John Doyle	Galway-Mayo Institute of Technology Department of Mechanical Engineering
Dr. Patrick Delassus MATERIAL	
PERANSI Y14.5	DRW0001



Π		
0		
	DESKGNER 28-Max-99 John Dayle SUPERVISOR	Galway-Mayo Institute of Technology Department of Mechanical Engineering Distanced Contents
	DESIGNER 28-Mar-99 John Dayle SUPERVISOR Dr. Pathick Delassus MATERIAL	Galway-Mayo Institute of Technology Department of Mechanical Engineering Dublin road, Galway TIRE Skeleton assembly







IX.

.

.

.



×



60		8.5•			
	DESIGNER 28-Mar 99 John Doyle Supervisor Dr. Patrick Dekassus	Gatway-M Deca	kayo Institute of Tec Iment of Mechanical Eng Dublin road, Galway	hnology (neering	
	DESIGNER 28-Mar 99 John Doyle Supervisor Dr. Patrick Dekassus MATERAI Fibre reinforced myon INTERPRET PER ANSI Y14.5	Gatway-M Deca TITLE S Pra/E Drawing	tayo Institute of Tec tment of Mechanical Eng Dublin read, Galway Tabilizer 1 Rite DRW000	hnology meeting	







XII

14


xiii

÷.



60]	8.5	
2	DESIGNER 28-Max-99 John Doyle SUPERVISOR Dr. Patrick Delassus	Galway-M Depar	ayo Institute of Technology Iment of Mechanical Engineering Dualin read, Galway
3	DESIGNER 28-Max-99 John Doyle SUPERVISOR Dr. Patrick Delassus MATERIAL Fibre reinforced nylon INTERPRET PER ANS	Galway-M Depar TITLE Pra/E Drawing	ayo Institute of Technology Iment of Mechanical Engineering Dualin rood, Galway tabilizer File



÷

8.5	
	Π
	-
DESIGNER 28-Mor-99	Galway-Mayo Institute of Technology
John Doyle	Department of Mechanical Engineering
Dr. Patrick Delassus	Dublin road, Golway
MATERIAL Fibre reinforced nylon	Stabilizer 2
MATERIAL Fibre reinforced nylon	Stabilizer 2 Proje Drawing File
MATERIAL Fibre reinforced myon INIERPRE: PER ANSI Y16.5	Prove Drawing File DRW0010





D			
-			
11			
A	120/2		
R	2000	S	
R	2000)	
	SS SS SS)	
	AC D)	
		>	
	DESIGNER 29 Mar-99 John Doyle	Catway-Mayo Institute of Technology Department of Mechanical Engineering	
	DESIGNER 29-Mar-99 John Doyle SuPERVISOR	Catway-Mayo Institute of Technology Department of Mechanical Engineering Dubin read, Gatway	



~	
D	
Ha	
2	
all a	
11-	
P	
Left	
DESIGNER 29-Mar-99	Galway-Mayo Institute of Technology
DESIGNER 29-Mar-99 John Doyle SUPERVISOR	Galway-Mayo Institute of Technology Department of Mechanical Engineering Dublin road, Galway
DESIGNER 29-Mar-99 John Doyle SUPERVISOR Dr. Patrick Delassus	Galway-Mayo Institute of Technology Department of Mechanical Engineering Duble road, Galway TRLE
DESIGNER 29-Mar-99 John Doyle SUPERVISOR Dr. Potrick Delossus MATERIAL See with mochany enome that INTERIAL See with mochany enome that	Galway-Mayo Institute of Technology Department of Mechanical Engineering Dualth road, Galway TRLE Front shackle
DESIGNER 29-Mar-99 John Doyke SUPERVISOR Dr. Patrick Delassus MATERIAL Sees with machinery enome their METERIAL Sees with machinery enome their PER AIVSI Y14.5	Galway-Mayo Institute of Technology Department of Mechanical Engineering Dublin road, Galway TRLE Front shackle Prafe Drawing File DRW0012

Appendix C Photographs



(a)





Figure 1 (a) Plan view of hydrofoil (without coloured gel coat) (b)Side view of hydrofoil.



Figure 2 Male form



Figure 3. Hydrofoil mould used to fabricate male form.



Figure 4. Male form use to fabricate final hydrofoil mould.



Figure 5. Male form used to fabricate final mould for nose section.



Figure 6. Part of conical tail



Figure 7. Machined male form.



Figure 8. Completed U-Kite



Figure 9. Front view of U-Kite

xxiii



Figure 10.

Appendix D

Mathematica Notebook/Program Listing (Lifting line theory).

The following Mathematica Notebook allows the user to define hydrofoil planforms and to define foil root and foil tip section properties. The program assumes a linear variation of section properties between wing root and tip and that the loading will be symmetric about the centre of the foil. The program uses the lifting line equations to determine the coefficient lift for a given angle of attack, and planform, the notebook will display the resulting distribution of section lift coefficient, and circulation across the span.

Clear[]

Wing Geometry Foilspan SP = 336; Chord at foil tip CHT = 160; Chord at foil root CHR = 160; Aspect ratio AR = (2 * SP / (CHT + CHR)) $\frac{21}{10}$ $Equivalent lift-curve slope ae, has been assumed to be equal to <math>2\pi$. $ae = 2 \times \pi$; Foil taper ratio

 $\lambda = CHT / CHR$

1

Spanwise coordinate

 $\phi = (22.5 \times \pi) / 180;$

A four-term series will be used to represent the spanwise loading.

Defining μ ,

where:

```
\mu_{n}aSin(nf) = A_{1}Sin(n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(3n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(n\phi)) + A_{2}Sin(n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(n\phi)) + A_{2}Sin(n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(n\phi)) + A_{2}Sin(n\phi)(\mu_{n}+Sin(n\phi)) + A_{2}Sin(n\phi)(\mu_{n
A_3Sin(5n\phi)(\mu_n + Sin(n\phi)) + A_4Sin(7n\phi)(\mu_n + Sin(n\phi))
 \mu_1 = (\operatorname{ae} \div (2 \times \operatorname{AR} \times (1 + \lambda))) \times (1 + ((\lambda - 1)) \times \operatorname{Cos}[\phi]);
 \mu_2 =
      (ae + (2 \times AR \times (1 + \lambda))) \times (1 + ((\lambda - 1)) \times Cos[2 \times \phi]);
 \mu_3 =
      (ae \div (2 \times AR \times (1 + \lambda))) \times (1 + ((\lambda - 1)) \times Cos[3 \times \phi]);
 \mu_4 =
       (ae \div (2 \times AR \times (1 + \lambda))) \times (1 + ((\lambda - 1)) \times Cos[4 \times \phi]);
Angle of attack, a
 \alpha = (-4 \times \pi) / 180;
 b[1, 1] = (Sin[\phi] \times (\mu_1 + Sin[\phi]));
 b[1, 2] = (Sin[3 \times \phi] \times ((3 \times \mu_1) + Sin[\phi]));
 b[1, 3] = (Sin[5 \times \phi] \times ((5 \times \mu_1) + Sin[\phi]));
 b[1, 4] = (Sin[7 \times \phi] \times ((7 \times \mu_1) + Sin[\phi]));
 b[2, 1] = (Sin[2 \times \phi] \times (\mu_2 + Sin[2 \times \phi]));
 b[2, 2] = (Sin[3 \times 2 \times \phi] \times ((3 \times \mu_2) + Sin[2 \times \phi]));
 b[2, 3] = (\sin[5 \times 2 \times \phi] \times ((5 \times \mu_2) + \sin[2 \times \phi]));
 b[2, 4] = (Sin[7 \times 2 \times \phi] \times ((7 \times \mu_2) + Sin[2 \times \phi]));
 b[3, 1] = (Sin[3 \times \phi] \times (\mu_3 + Sin[3 \times \phi]));
 b[3, 2] = (Sin[3 \times 3 \times \phi] \times ((3 \times \mu_3) + Sin[3 \times \phi]));
 b[3, 3] = (Sin[3 \times 5 \times \phi] \times ((5 \times \mu_3) + Sin[3 \times \phi]));
 b[3, 4] = (Sin[3 \times 7 \times \phi] \times ((7 \times \mu_3) + Sin[3 \times \phi]));
 b[4, 1] = (Sin[4 \times \phi] \times (\mu_4 + Sin[4 \times \phi]));
 \mathbf{b}[\mathbf{4}, \mathbf{2}] = (\mathbf{Sin}[\mathbf{4} \times \mathbf{3} \times \phi] \times ((\mathbf{3} \times \mu_4) + \mathbf{Sin}[\mathbf{4} \times \phi]));
 b[4, 3] = (\sin[4 \times 5 \times \phi] \times ((5 \times \mu_4) + \sin[4 \times \phi]));
 b[4, 4] = (Sin[4 \times 7 \times \phi] * ((7 \times \mu_4) + Sin[4 \times \phi]));
 B = Array[b, \{4, 4\}] // MatrixForm
```

xxvii

```
0.432693 2.42673 3.80885 2.15017
1.02891 2.08674 -3.14457 -4.2024
1.54461 -1.21229 -1.78479 5.69098
1.748 -3.24399 4.73999 -6.23599
```

lud = LUDecomposition[B]

{{{1.748, -3.24399, 4.73999, -6.23599},
{0.588625, 3.99624, -5.93465, -0.531747},
{0.247536, 0.808195, 7.43189, 4.12356},
{0.883647, 0.413953, -0.473177, 13.3727}},
{4, 2, 1, 3}, 7.09491}

$c[1, 1] = \alpha \times \mu_1 \times Sin[\phi]$

-0.0199838

$c[2, 1] = \alpha \times \mu_2 \times Sin[2 \times \phi]$

-0.0369252

$c[3, 1] = \alpha \times \mu_3 \times Sin[3 \times \phi]$

-0.0482451

$\mathbf{c}[\mathbf{4}, \mathbf{1}] = \alpha \times \mu_4 \times \mathbf{Sin}[\mathbf{4} \times \phi]$

-0.0522201

 $Cx = Array[c, \{4, 1\}] / MatrixForm$

```
-0.0199838
-0.0369252
-0.0482451
-0.0522201
```

a = LUBackSubstitution[lud, Cx];

N[a] // MatrixForm

-0.0329052 -0.00193277 -0.000255488 -0.0000383662

Define A_1

A[1, 1] = N[Rationalize[Take[a, 1], 10^-10]];

Drop[a, 1];

Define A_2

A[2, 1] = N[Rationalize[Take[%, 1], 10^-10]];

xxviii

Drop[a, 2];

Define A_3

 $A[3, 1] = N[Rationalize[Take[\%, 1], 10^{-10}]];$

Drop[a, 3];

Define A₄

A[4, 1] = N[Rationalize[Take[%, 1], 10^-10]];

Construct column matrix S containing the coefficients $A_1, A_2, A_3 \& A_4$

S = Array[A, {4, 1}] // MatrixForm

(-0.0329052) (-0.00193277) (-0.000255488) (-0.0000383661)

 $Plot[{A[1, 1] Sin[x] + A[2, 1] Sin[3x] + }$

A[3, 1] Sin[5 * x] + A[4, 1] Sin[7 * x],

{x, 0, π}, Frame -> True, GridLines -> Automatic, PlotRange -> All, PlotLabel -> "Spanwise circulation", Background -> RGBColor[0, 1, 1]];





Plot[{A[1, 1] Sin[x]}, {x, 0, π}, Frame -> True, GridLines -> Automatic, PlotRange -> All, PlotLabel -> "Spanwise lift distribution", Background→ RGBColor[0, 1, 1]];



Figure 4.17 Symmetric spanwise lift

Coefficient of lift $C_L = A[1, 1] \times \pi \times AR$

 $\{\{-0.217087\}\}$

Coefficient of lift induced drag

$$C_{DV} = \left(\frac{Cl^{2}}{\pi \times AR}\right) \times \left(1 + \frac{3 \times A[2, 1]^{2}}{A[1, 1]^{2}} + \frac{5 \times A[3, 1]^{2}}{A[1, 1]^{2}} + \frac{7 \times A[4, 1]^{2}}{A[1, 1]^{2}}\right)$$

$$\{(0, 00721945)\}$$