STUDY OF VORTEX TRAP ON LOW ASPECT RATIO WING WITH NACA 0015 CROSS SECTION

SULIMAN ABD.Y.GANDOUZ

A project report is submitted of the fulfillment of the requirements
for the award of the degree of Master of Mechanical
Engineering (Thermofluid)

Faculty of Mechanical Engineering and Manufacturing
University Tun Hussein Onn Malaysia (UTHM)

MAY 2011

ABSTRACT

The purpose of this project is to apply a numerical procedure for flow analysis over a low wing aspect ratio with and without a groove by Gambit and FLUENT software. Gambit software used for generating mesh over flow domain surrounding the wing, while FLUENT as the flow solver software. The wing has a rectangular form with its airfoil cross section is NACA 0015 and wing aspect ratio equal to 0.66. Numerical procedure starts with creating wing plan form by used solid work software. The data geometry then transported to Gambit in order to allow mesh flow domain can be created. This had been done on two wing models. The first wing model is considered as the wings with clean airfoil section while the second one as the wing with its airfoil with groove. The first wing model is called as a baseline model. Both models then solved by use of Fluent software at the free stream velocity at 40 m/sec and Reynolds number R_L=400000 for different angle of attacks at $\alpha = 0^0$, 5^0, 15^0, 25^0 and 40^0. The result shows that in term of overall aerodynamics characteristics, namely the lift coefficients C_L, had been found relatively low compared to the typical value of lift coefficient of the airfoil. This result represents a reasonable result since typical value of lift coefficient for a low aspect ratio is very low compared with the lift coefficient of its airfoil.

The consequence of low aspect ratio made the wing able to delay the presence of flow separation although setting angle of attack is high. As result the effect groove in this present work in term of delaying the flow separation is so significant. Wing without groove still able to produce lift forces as it happen on the wing with groove. The comparison result flow phenomena between those two wing models are discussed. The flow analysis for the same wing model but with high aspect ratio is suggested for the future work in order to give a better assessment for the influence of groove.

CHAPTER 1

STUDY OF VORTEX TRAP ON LOW ASPECT RATIO WING WITH NACA 0015 CROSS SECTION

1.0 Introduction

Flight has been a major part of the world since it was first demonstrated by the Wright brothers in 1902. However, in depth studies into the effects of airflow over wings didn't occur until World War I (Anderson). In an attempt to better understand what made a good wing, the National Advisory Committee for Aeronautics, henceforth referred to as the NACA, was founded. In 1933 .
Aerodynamic design has seen a rise in the implementation of multifunctional devices and actuators that allow for dramatic changes in performance with only slight variations to the effective surfaces. Although rote airfoil design has essentially yielded its peak in performance, auxiliary mechanisms are being investigated and explored for their potential in making airfoils more functional, especially in demanding environments such as edge-of-the envelope performance, unmanned light and fast, and high-lift low-speed applications.

In this preliminary study, it will use cfd software to enable study on the flow behaviour surrounding the airfoil. Using cfd, one can build a computational model that represents a system or device under study. Then, apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore cfd is a sophisticated computationally-based design and analysis technique. cfd software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics of through computer modelling. Using cfd software, you can build a 'virtual prototype' the system or device that you wish to analyze and then apply real-world physics and chemistry to the model, and the software will provide you with images and data, which predict the performance of that design (Wenbin Song, Andy Keane, Hakki Eres, Graeme Pound, and Simon Cox Two Dimensional Airfoil Optimisation Using CFD in a Grid Computing Environment)

By using cfd software in this preliminary study explores the possibility of trapping single and multiple vortices on an airfoil. The vortex always moves around its equilibrium position. A boundary with multiple corrugations can be created and incorporated onto an airfoil to trap multiple vortices

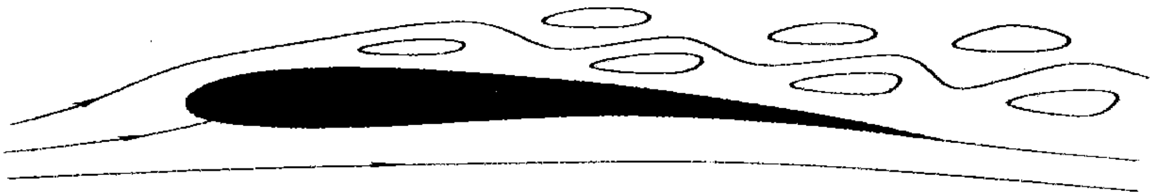

Figure 1.1 vortex shedding from a generic airfoil

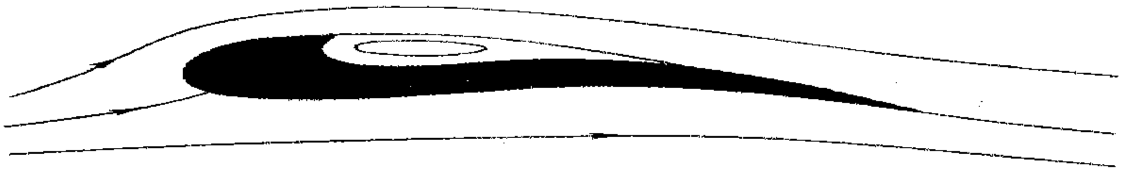

Figure 2.1 airfoil with a cavity an trapped vortex

That will improve to increase the lift coefficient C_L

1.1 Characterizing Airfoils

Before we develop a model for the fluid flow around airfoils, it is important to define air-foils geometrically and to acquaint ourselves with the nomenclature with which they are characterized.

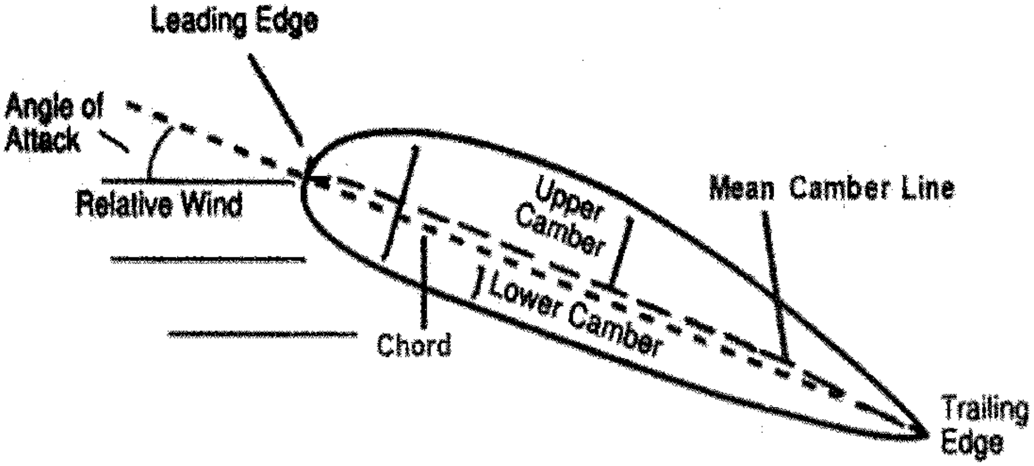

Figure 1.3: Diagram of an airfoil with key parameters labelled

The above diagram labels the main parameters of an airfoil that play a key role in its aerodynamic performance. Specifically, we are interested in the angle of attack the chord length, and the mean camber line. The chord is a straight line typically used to measure airfoil length, whereas the mean camber line is a curve halfway between the upper and lower surfaces used to measure airfoil curvature.

Airfoil shapes are commonly characterized with a numbering system originally defined by the National Advisory Committee for Aeronautics (NACA). This characterizing system defines airfoil shapes with a series of digits corresponding to non-dimensionalized airfoil properties. The number of digits used to describe an airfoil corresponds to the complexity of the airfoil. For this project we will only model four digit (4-series) airfoils to simplify the geometry of the airfoils we wish to analyze. In 4-series airfoils, the profile is defined as follows:

1. The first digit describes the maximum camber as a percentage of the chord length. The maximum camber is the maximum distance between the chord and mean camber line along the axis of the chord.
2. The second digit indicates the position of the maximum camber in tenths of the chord.
3. The last two digits provide the maximum thickness of the airfoil as a percent of the chord length .

For example, the NACA 2412 airfoil has a maximum camber of 2% located 40% (0.4 chords) from the leading edge with a maximum thickness of 12% of the chord. Four-digit series airfoils by default have maximum thickness 30% of the chord (0.3 chords) from the leading edge .

The NACA 0015 airfoil is symmetrical, the 00 indicating that it has no camber. The 15 indicates that the airfoil has a 15% thickness to chord length ratio: it is 15% as thick as it is long .

1.2 Airfoil Aerodynamics

Airfoils are two-dimensional wing sections or "lifting-surfaces." The resultant forces and moments acting on an airfoil are the net result of the action of the distributed pressure around the airfoil surface and viscous shear forces at the surface. Figure 1.6 shows the pressure and shear forces acting on an element of the airfoil surface. The resultant force and moment are obtained by integrating the elemental pressure and shear forces around the airfoil. These forces can be resolved into a chord-axis system, with normal and axial forces, or a wind-axis system, with lift and drag forces.

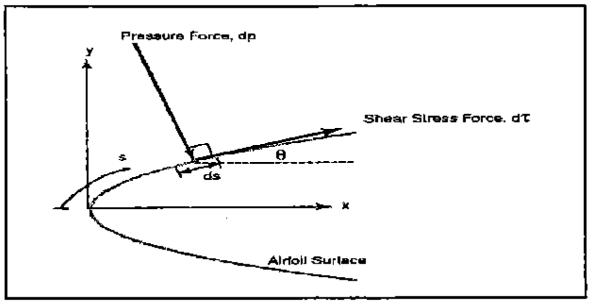

Figure 1.4: Pressure and shear forces acting on an element of the airfoil surface (Leishman, J. Gordon. 2000)

A schematic of the decomposition of these forces on an airfoil is shown in Figure 1.7.

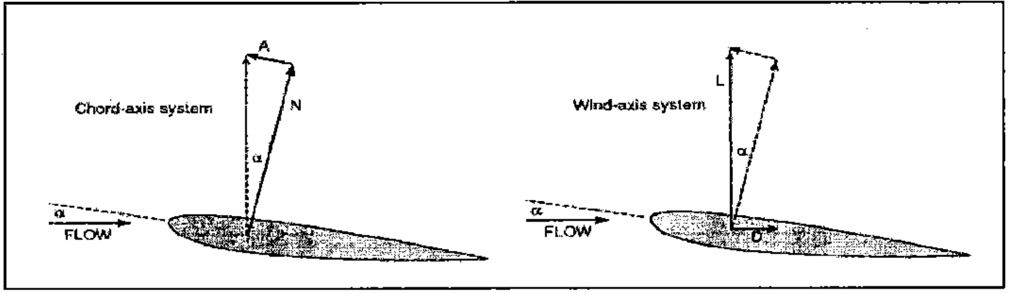

Figure 1.5: Schematic of the decomposition of resultant forces on airfoil. Adapte from Leishman (Leishman, J. Gordon. 2000)

For the normal and lift forces, the pressure force dominates, and the surface shear force contribution is negligible. For the axial and drag forces, the shear stress contribution has a measurable effect and must be taken into account. The surface pressure is typically presented in terms of a pressure coefficient, C_p In compressible flow, the definition of the coefficient of pressure comes from Bernoulli's equation.

$$P_\infty + \frac{1}{2}\rho_\infty U_\infty^2 = P + \frac{1}{2}\rho_\infty U^2 \quad (1.1)$$

Where P and U are the local pressure and velocity and P_∞, ρ_∞, and U_∞ refer to the pressure, fluid density, and velocity in the free stream. The pressure coefficient is defined as the difference between local and free stream pressure divided by the dynamic pressure in the freest ream.

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2} \quad (1.2)$$

Figure 1.8 illustrates a qualitative comparison of the pressure distribution over an airfoil at a high angle of attack with and without the flow separating at the leading edge. When the flow separates, the pressure distribution on the bottom surface does not change. However on the upper surface, the separated flow results in a higher absolute pressure so that the difference between the upper and lower surface is less. Therefore, the lift is lower when the flow separates. In addition to the loss of lift, there is an increase in drag caused by flow separation, so the lift-to-drag ratio decreases significantly. A summary of airfoil characteristics are compiled by Abbott and von Doenoffjt. These include lift coefficients and drag polar versus angle of attack up to and past where separation occurs.

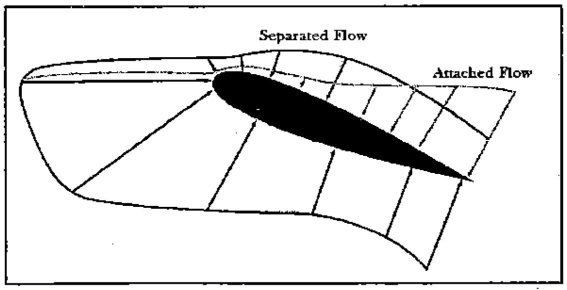

Figure 1.6: Qualitative comparison of the pressure distribution over an airfoil. Adapted from Anderson(Anderson Jr., J.D. 1989)

1.3 Problem Statement

The effective angle of attack near the centre of rotation of the rotor blade is normally high. As result the lift at that section may lose or drop significantly due to flow separation. This flow separation occurred if the flow inside the boundary layer does not enough kinetic energy to overcome the positive pressure gradient in its downstream direction. If such flow separation can be avoided or delayed, then the lost of lift at that section can be eliminated. The vortex trap approach may represent one of the methods to maintain the lift which will be investigated in the present study.

1.4 Objectives of study

The primary objective of the study is to improve lift coefficient C_L of a NACA 0015 airfoil at high angle of attack by exploring the possibility of trapping multiple vortices on a surface of airfoil by incorporating grooves.

1.5 Scopes of study

1. To make the computer model 3-D of the wing with NACA 0015.
2. To do analysis of the pressure distribution on top and bottom airfoil.
3. Calculate the lift coefficient C_L.
4. All the above tests at high angle of attacks (0, 5, 15, 25, and 40) degree.
5. For one vortex trap.
6. Compare between the Clean wing with NACA0015 (baseline) and a modified Wing with NACA0015-TV having a trapping cavity on its upper surface.

1.6 Expected Result

1. The results from the study will show us that an airfoil can be stabilized using vortex trap.

2. The use of one or more than one vortex traps for effective active vortex control will depend on the results.

CHAPTER 2

LITERATURE REVIEW

2.0 INTRODUCTION

Tip vortices are generated at the tip of lifting surfaces where fluid flows from the high-pressure side to the low-pressure side, then rolls up and travels to the blade wake. The tip vortex system originated from the complex three-dimensional separated flow is highly unsteady and turbulent. The interactions between the primary and the secondary vortices, the vortices and the separated shear layer, and the vortices and the wakes occur simultaneously in the flow field.

The wing tip vortex is of importance because of its effects on many practical problems such as the landing distances for aircraft, the blade/vortex interactions on helicopter blades, propeller cavitations on ships, and other fields. For example, tip vortices contribute to the induced drag of the generating surface, a situation that is exacerbated for low aspect ratio surfaces such as marine propellers. When vortices shed from the helicopter rotor blade interact with a following blade, the resulting unsteady forces may contribute to the premature blade failure. Blade vortex interaction also had been identified as the major source of noise generated by a rotor. In the airport, the trailing wing tip vortices produced by the large transport aircrafts pose a hazard to smaller following aircrafts. Therefore, in order to minimize the separation time of aircraft during takeoff, as well as to reduce the tip–vortex-induced drag and noise, it is absolutely necessary to study and predict or control the wing tip vortex, so as to allow the most efficient use airport facilities and improve the performance of blade and propellers.

Numerous experimental investigations have been conducted to understand the tip vortex structure and its dissipation or persistence. Shekarriz et al studied the evolution of the blade tip vortex by mapping its instantaneous lateral velocity at several consecutive axial locations(A. Shekarriz, et al 1993). They also measured the axial velocity distribution. Lavi Zuhai et al carried out experimental investigations on blade tip vortex(Lavi Zuhai et al 2001), They found that the blade sheds multiple vortices. The interaction of the vortices gives rise to the unsteady motion of the blade tip vortex. Andreas Vogt et al presented their PIV measurement results on the tip vortex of a NACA0012 blade (Andreas Vogt et al)The velocity profiles indicate a non linear increase of the azimuthal velocity with growing distance from the vortex center as it is implied by the solid body rotation model of the vortex core. Devenport et al performed experiments on the tip vortex generated from a rectangular NACA 0012 half-wing (Devenport, et al 1996) A very important conclusion of their work is that the vortex is laminar. Chow, Zilliac and Bradshaw took measurements of a blade tip vortex (Chow, et al 1997) They found that the turbulence in the tip vortex decayed quickly along stream-wise direction. Spalart gave a review on the wingtip vortex (Philippe R. Spalart 1998)

Elgin Anderson at el found the relationship between the vortex strength and axial velocity in a trailing vortex(Anderson et al 2003) However, their results indicated that the magnitude of the axial velocity is sensitive to the two end-cap configurations (flat and rounded). E.A. Anderson at el studied the influence of wing end-cap geometry and flow parameters on the wingtip vortex (Anderson 2001) There are also some experiments on controlling wing tip vortex. Charles S. Matthewson et al carried out an experimental study on the effects of the destabilizing the tip vortices from a model rotor blade using perturbations introduced by discrete jets located at the tips (Charles et al) Joshua et al performed the aerodynamic analysis on the potential of wing tip sails to control the wing tip vortex and increase the aerodynamic efficiency(Joshua B. Et al 2001). There are also many numerical simulations for wingtip vortex based on the Reynolds averaged Navier-Stokes equations (Pankajakshan, R., Taylor et al 2001) Dacles-Mariani, et al, gave a RANS investigation on wingtip vortex based on the experimental boundary data (Dacles-Mariani.et al 1995)

2.1 Literature review

The idea of trapping a vortex is old, and it is not easy to find out who first suggested it (Raffaele S. Donelli et al 2010) A trapped vortex device is a cavity to trap flow vortices to reduce pressure drag levels past bluff bodies and/or thick wing airfoils. Preliminary studies showed the need to apply constant mass flow suction inside the cavity to stabilize the vortex. The aim of this work is to find the best location and the minimum suction distribution to be applied in the trapped vortex mounted on a NACA0024 airfoil to be tested in the 3 diameter wind tunnel section of the Politecnico of Turin in order to delay flow separation. This activity has been carried out in the framework of the European project VCELL2050 A CFD parametric study has been performed to analyze the flow characteristics of a cavity equipped airfoil with vortex trapping and suction playing. Different values of the suction mass flow rate and of the suction slot locations have been considered.

A comparison between the airfoil shape with no cavity + distributed suction and with trapped vortex cavity + suction has been carried out. The trapped vortex cavity resulted to be more effective with respect to the distributed suction either in terms of lift/drag coefficient either in terms of aerodynamics efficiency/required energy for the control. On the other hand, the cavity with distributed suction (SALL) was found to be the best compromise in terms of aerodynamic performances and energy or power needed to realize the control when compared to the S3 configuration. It has to be stressed that some margins of improvement in the suction location can be still foreseen: this could be numerically achieved through a real optimization process aiming at searching the optimal suction locations and mass flow rates which allows for the minimum value of the energy due to the suction system under fully attached flow/low pressure drag constraint.(D. R. Troolin, et al 2005) A NACA 0015 airfoil with and without a Gurney flap was studied in a wind tunnel where Rec = 2.1 x 105 in order to examine the evolving flow structure of the wake through time-resolved PIV and to correlate this structure with time-averaged measurements of the sectional lift coefficient. The Gurney flap is a tab of small length (1% to 4% of the airfoil chord) that protrudes 90° to the chord at the trailing edge. The Gurney flap increases the lift on an airfoil while increasing the drag only minimally for cases where the height of the flap is within the boundary layer region. Multiple vortex shedding modes were seen upstream and downstream of the Gurney flap. The Gurney flap is a device that increases the lift on an airfoil while inducing only a limited amount of drag, adding an overall gain to the airfoil design. While it is somewhat intuitive that the Gurney flap increases lift by adding to the effective camber of the wing, the less-obvious advantage of the Gurney flap lies in the intriguing interaction of the counter-rotating vortices that are alternately shed downstream, as well as the vortices being shed from the cavity upstream of the Gurney flap tip.

The symmetric airfoil comes to a point at the trailing edge and leaves a weak and narrow wake. The Gurney design requires the turbulent flow downstream of the airfoil to follow a fairly structured set of paths which are the alternating counter-rotating vortices. Further, boundary layer fluid in the cavity upstream of the Gurney flap is trapped and intermittently released into the wake which in turn induces a net negative normal velocity on the airfoil wake increasing the circulation, and thus the lift. While the Gurney flap did not increase the stall angle of incidence, the flow it induced did serve to increase the lift at every angle of incidence measured .

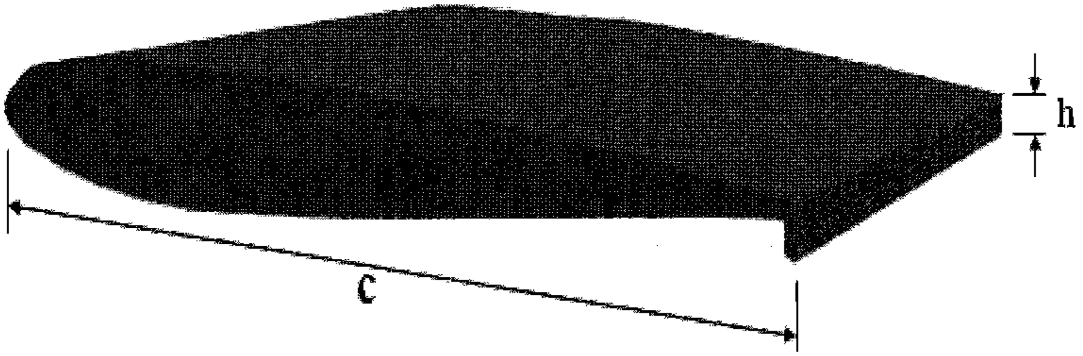

Figure 2.1: Gurney flap on an airfoil

(W.W.H. Yeung 2009) This paper explores the possibility of trapping single and multiple vortices on a surface and on an airfoil. Results from the vortex trajectory show that an indented surface and an airfoil with an indentation can stabilize a trapped vortex. That is, when displaced by a finite distance from the equilibrium position, the vortex always moves around its equilibrium position. A boundary with multiple corrugations can be created and incorporated onto an airfoil to trap multiple vortices. Irregular vortex motion has been found when multiple vortices are presented above the corrugations.

A method is proposed, allowing the vortices to spiral towards its equilibrium positions. It has been demonstrated in this paper that an indented surface is capable of trapping a stable vortex, when a flat surface fails to do so. Such a surface can be incorporated onto an airfoil by using conformal mapping. The present study also indicates that multiple vortex trapping is possible, although chaotic motion of vortices is found. If the vortex is replaced by a source and vortex, the chaotic motion can be suppressed, and the region of trapping a vortex is corresponding enlarged.

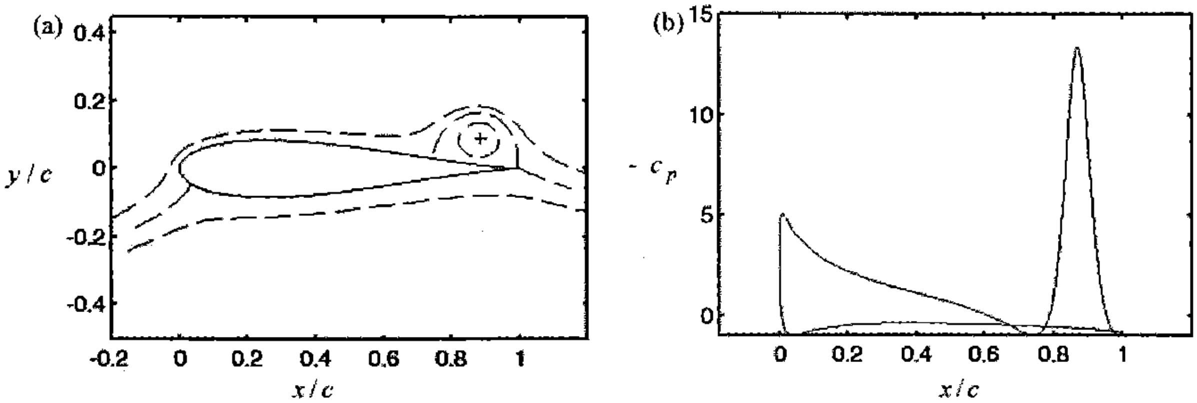

Figure 2.2 : (a) The recomputed streamline pattern and (b) the pressure distribution on a Joukowsky airfoil with a trapped vortex on the upper surface near the trailing edge based on an example from [ ]. (a) +: location of the stationary vortex, —: airfoil, ——: streamline

(Fabrizio De Gregorio et al 2008) This paper summarises the experimental campaign performed at CIRA CT-1 wind tunnel aimed to investigate the potential benefit obtainable using a trapping vortex cell system on a high thickness airfoil with and without steady suction and/or injection mass flow. The behaviour of a 2D model, equipped with a span wise oriented circular cavity, has been investigated. Pressure distribution on the model surface and inside the cavity and the complete flow field around the model and inside the cavity have been measured. An extensive test campaign has been carried out in the CT-1, an open circuit wind tunnel, with test section size of 305x305x600 mm3 and maximum speed of 55 m/s. Due to the limited dimensions of the WT, the model has been mounted on the bottom wall of the wind tunnel in order to avoid blockage problems. The model represents a two dimensional high thickness airfoil with a chord length of 350mm. The model angle of attack ranges between 5.66° to 12.66° with a step of 1°. The installation of the model on the wind tunnel bottom wall presented heavy flow instability under the front part of the model. The flow instability has been solved applying a flow suction in front the model trough a porous wall installed on the bottom WT wall.

The cavity has been realized with transparent material in order to allow optical access and consequently PIV measurements. The model has been designed in order to permit flow suction and/or blowing inside the cavity. The influence of different parameters has been investigated. Tests have been performed varying the wind tunnel speed (form 15 m/s, to 30 m/s), varying the suction mass flow (from 0 m3/h to 25 m3/h) varying the blowing mass flow (from 0 m3/h to 50 m3/h) applying suction and blowing at the same time, and varying the model angle of attack (AoA). In the paper the performed test campaign, the adopted experimental set-up, the data post-processing and the results' description are reported. An extensive test campaign has been successfully performed in order to investigate the potentiality of trapped vortex cavity flow control.

The TVC has been investigated as passive and active method, i.e. without and with mass transport. The full pressure and flow characterization has been detected for the base clean airfoil and for the model modified for carrying the flow control device. A complex experimental set up has been realised composed by different elements: the suction system for removing the unsteady circulation region in front the model, the suction/injection systems for active flow control in the cavity, the measurement system for recording all the boundary condition. From the showed results the following conclusions can be draw. Passive TVC flow control is not able to control the flow separation. The vortex is not confined in the cavity and vortex shedding is present decreasing the aerodynamic characteristics of the original airfoil. Active TVC flow control is able to control the flow separation, for limited values of the blowing coefficient full reattachment has been obtained. These first experiments provided really encouraging result. The campaign shall provide unique experimental data to the numerical colleague for code validation.

(A. bouferrouk et al 2005) In this study we investigate the effectiveness of some control techniques, both passive and active, for the stabilisation of a large-scale trapped vortex of a Lighthill's airfoil. The flow is two-dimensional, incompressible and inviscid solved using a discrete vortex method code. It was found that stabilisation improves the aerodynamic characteristics of the airfoil with active control achieving stabilisation with less energy input A Lighthill's airfoil with a cavity and strong steady suction is capable of stabilising a large-scale vortex thereby enhancing its aerodynamic performance. However, this is only possible up to a critical suction rate. The alternative of unsteady suction provides stabilisation with a reduced suction rate but may be limited in practice by actuator performance. The trapped vortex stability, defined with respect to large-scale vortex shedding, was proved using a simple exponential decaying model. The use of a linear feedback controller based on a stabilising parameter G was effective in retaining the trapped vortex stability with a reduced suction rate compared to the passive suction schemes. Achievement of this, however, seems to be dependent on the cavity shape. Application of vorticity flux control concept is another viable way of stabilising a trapped vortex using dynamic flow rates of suction/blowing. Although stabilisation was not achieved with reduced flow rate compared to continuous suction alone, active flux control achieves stabilisation with decreased drag. This represents a potential saving in energy. There remain open questions on the optimisation of such stabilisation approaches and power balance requirements.

CHAPTER 3

METHODOLOGY

3.0 INTRODUCTION

Computational fluid dynamics (CFD) has been constantly developed over the past few decades and now both commercial and in-house codes can provide more and more robust and accurate results. Combined with the use of wind tunnel test data, CFD can be used in the design process to drive geometry change in- stead of being used mainly as a design validation tool. This aspect can be further exploited by bringing optimisation tools into the design process. Automation of the design process can significantly shorten the design cycle and lead to better designs compared to previous manual design modification approaches. Such manual approaches are still adopted by most engineers due to various reasons: lack of robustness and flexibility in automating the design process, the high computational cost associated with large numbers of iterations of high fidelity simulation codes, difficulties of collaborations in a heterogeneous computational environments, etc.

In fact, the revolution brought by the World Wide Web with respect to information sharing has not yet delivered fundamental changes to engineering design practice for a number of reasons, including security problems in collaborative environments. The emerging Grid computing technologies (Wenbin Song et al)

3.1.1Airfoil Configuration

The airfoil chosen for the present work is NACA0015 (symmetric and having a maximum thickness of 15 %). The airfoil has a chord length of 150 mm and maximum thickness of 22.5 mm as shown in figure 3.4.

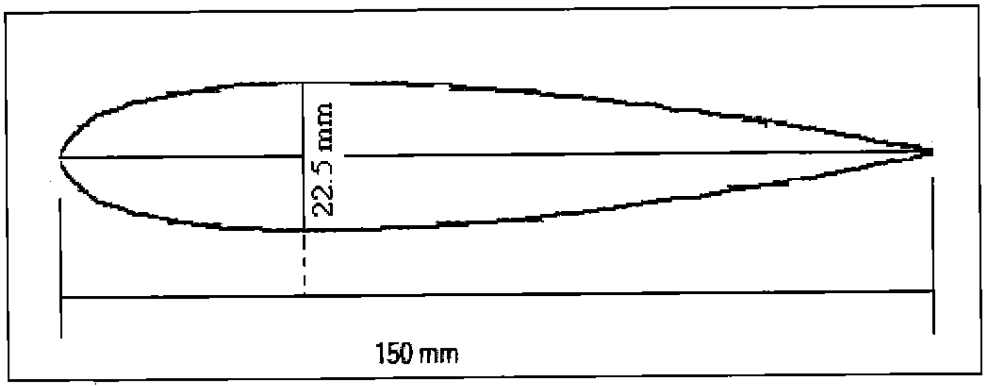

Figure 3.1: Sketch of an airfoil NACA 0015 (the maximum Thickness at 30% of chord)

3.1.2Airfoil nomenclature

Consider the blade of helicopter, as sketched in Fig. 3.2. The cross-sectional shape obtained by the intersection of the blade with the perpendicular plane shown in Fig. 3.2 is called an airfoil. Such an airfoil is sketched in figure 3.3, which illustrates some basic terminology (john D. Anderson, Jr. 2008)

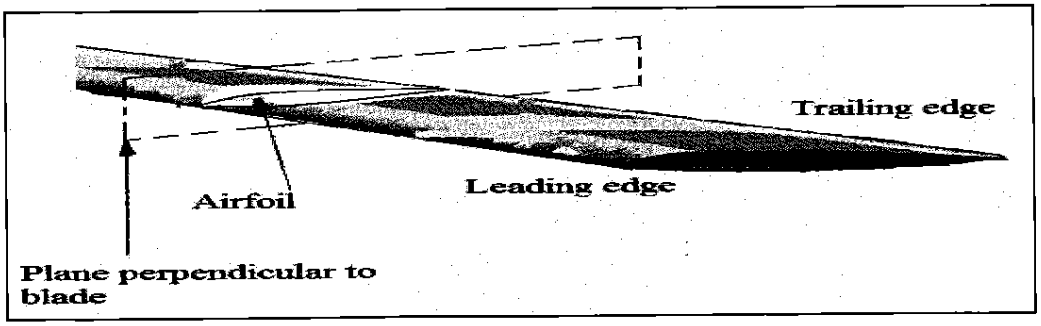

Figure 3.2: Sketch of blade and an airfo

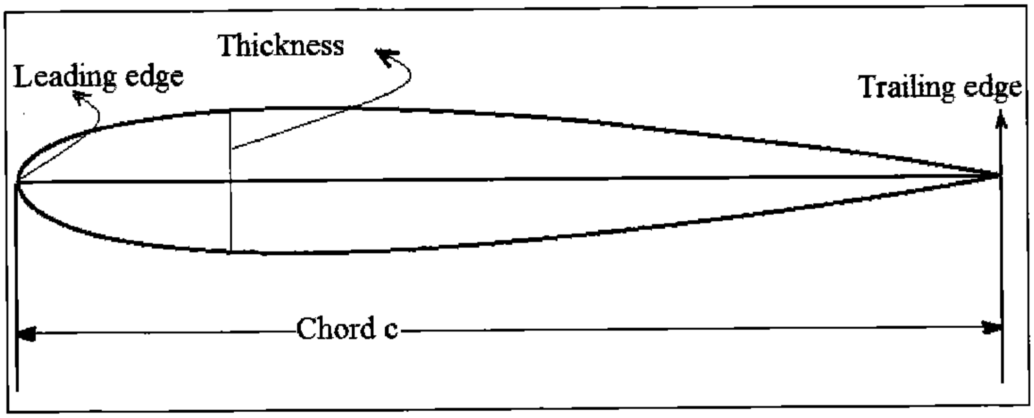

Figure 3.3: Airfoil nomenclatures, the shape shown here is a NACA 0015 airfoil

The straight line connecting the leading and trailing edges is the chord line of airfoil, and the precise distance from the leading to trailing edge measured along the chord line simply designated the chord of the airfoil, given by the symbol c.

3.2 Flow Chart

Figure3.1 below shows the methodology procedure in a flow chart

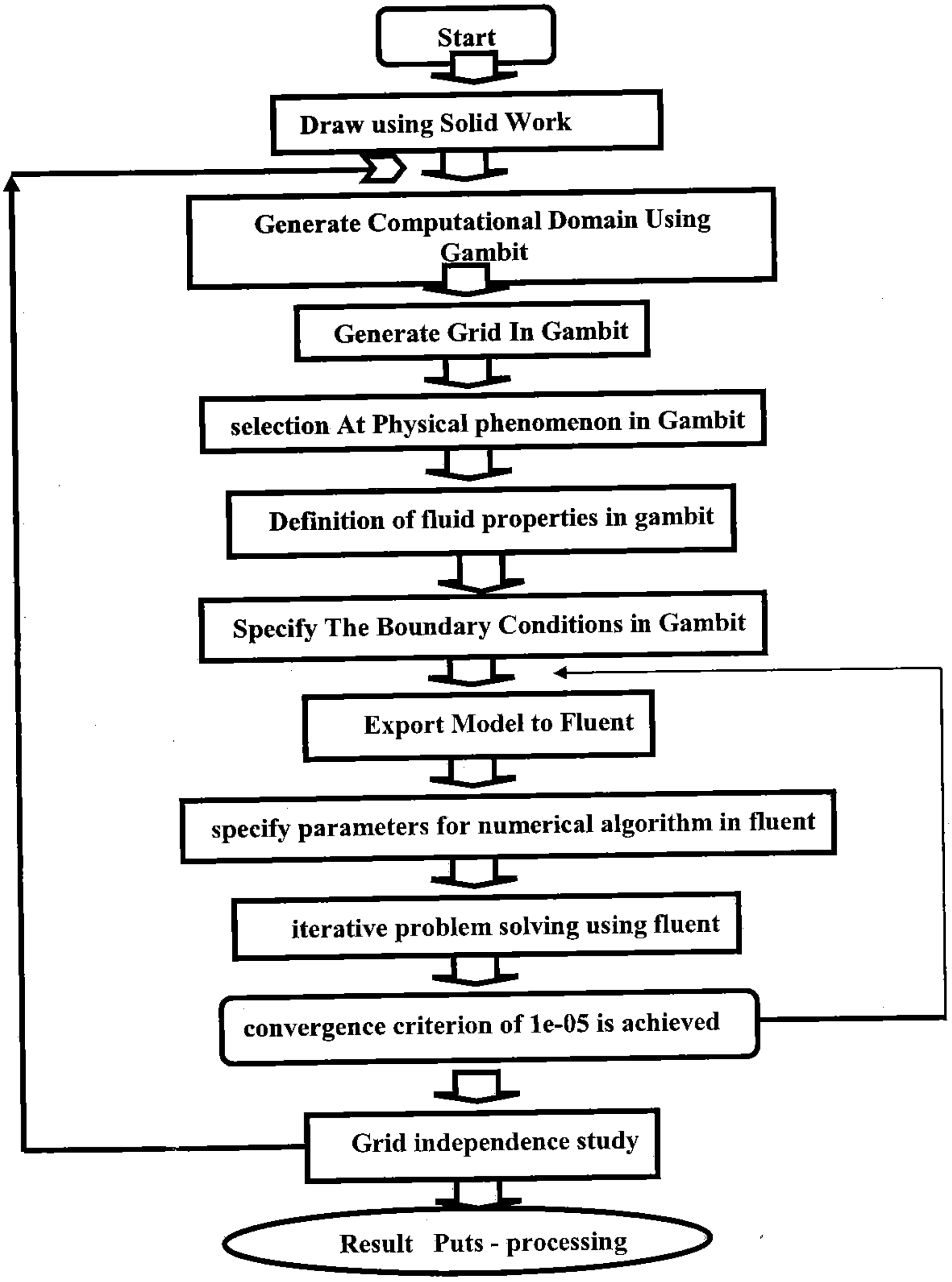

Figure 3.4 computational domain and geometry

3.3 Computational Domain and Geometry

The geometry and the computational grid was generated using (GAMBIT 2.3.16) and The commercial CFD code FLUENT (FLUENT, 6.3.26) will use to analyze the model flow characteristics of Clean wing with NACA0015 (baseline) and A modified NACA0015-TV wing having a trapping cavity on its upper surface, The objective is to demonstrate the feasibility and the effectiveness of the trapped vortex to reduce pressure drag and to increase lift. Modeling and mesh generation were however performed in Gambit environment .the air has been taken as the fluid flow on both airfoils . table 3.1 show the schematic diagram of NACA0015 airfoil The blade model were made from arcylate material (polymer), the dimensions of airfoil for test will be 150 mm chord and maximum thickness 22.5mm and the Width is100mm as shown in figure 3.1 .

The objective is to demonstrate the feasibility and the effectiveness of the trapped and drag measurements will be performed on the clean airfoil, as well as, on the wing equipped with a trapped vortex cavity successively the angle of incidence (0, 5, 15, 25 and 40) degree and 40 m/s air speed.

Table 3.1 The dimension of the wing with NACA0015 geometry parameter.

Width of the wing	**Chord of length**	**maximum thickness**	**Angle of attack(AOA)**	**air speed**
100mm	150 mm	22.5 mm	(0, 5, 15, 25 and 40) degree	40 m/s

3.4 Fluid and material Conditions

CFD analysis will be horizontal for both airfoils and flow is turbulent. Thermal properties of air at 1 atm pressure (101,325 Pa) and 293.15 K = 20±C = 68±F, similar to Thompson, as shown 3.2. The type of material airfoils are will be using in this analysis is arcylate material (polymer) and the properties are show in table 3.3.

Table 3.2 the properties for air

Temperature - *t* - (°C)	Density - *ρ* - (kg/m^3)	Specific heat capacity - *c$_p$* - (kJ/kg K)	Thermal conductivity - *l* - (W/m K)	Kinematic viscosity - *ν* - (m^2/s) x 10^{-6}
20	1.205	1.005	0.0257	15.11

Table 3.3 the properties for arc late material

Type of material	Chemical name	Specific gravity	thermal conductivities	Max. Operating temp. (°C)
Acrylic	polymethyl methacrylate	1.18	0.16 to 0.45 W/m/K	55

3.4.1 Pressure coefficient

The Pressure Coefficient tells the engineer how the air is acting as it travels around an airfoil. When the air slams into the leading edge of the airfoil, its speed goes to zero. This makes the pressure coefficient equal to 1. As the air moves along the sides of the airfoil and speeds up, the Pressure Coefficient becomes more and more negative. A really negative Pressure Coefficient corresponds to low pressure, or suction. Most airfoils have the most negative Pressure Coefficient on the upper surface. That is why an airfoil is sucked up into the airflow. The pressure coefficient plot represents the data used to obtain important number which is lift coefficient.

3.5 Methodology steps

3.5.1 Draw using SolidWork

The models used for this study were designed using the CAD program SolidWorks, they were two models of wings the first one was without groove and second one was with groove ,the both of wings same scales at 150 mm chord and maximum thickness 22.5mm and the Width is100mm as shown in the figure 3 a for with groove and figure 3 b without groove

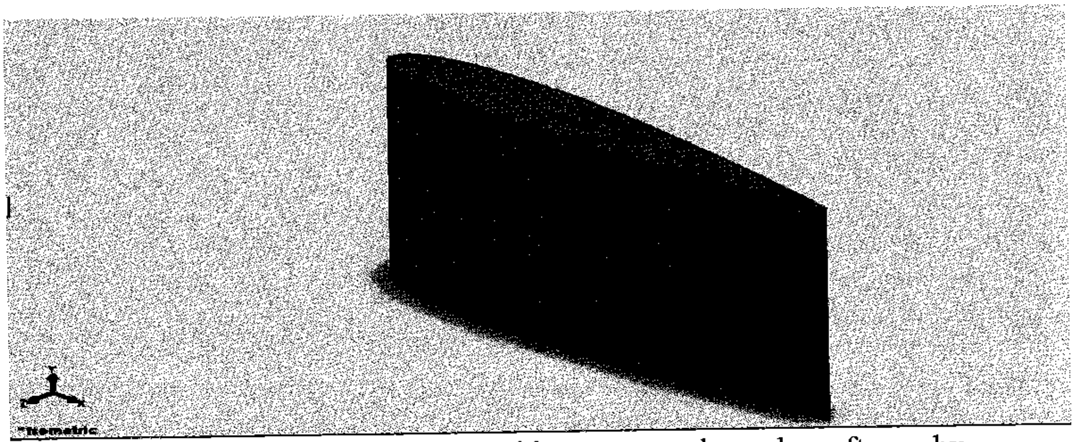

Figure3.5 a the wing with NACA0015 without groove drawn by software by solidworks

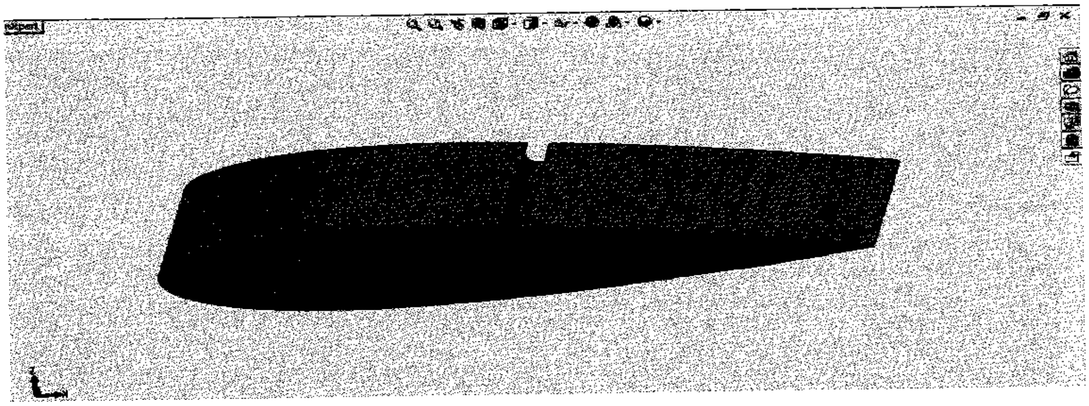

Figure3.5 b the wing with NACA0015 without groove drawn by software solidwork

3.5.2 Outer mesh boundary

.Numerical procedure starts with creating wing plan form by used solid work software. The data geometry then transported to Gambit in order to allow mesh flow domain can be created. This had been done on two wing models. The first wing model is considered as the wings with clean airfoil section while the second one as the wing with its airfoil with groove. and great outer mesh boundary for both the wings As in the figures 3 a and 3 b below

figure 3.6 a the wing with groove and without groove in gambit software

Figure 3.6b shown Outer mesh boundary around the wing