ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL OF SCIENCE ENGINEERING AND TECHNOLOGY

PUMPING THE MAIN SAIL FOR PERFORMANCE GAIN

M.Sc. THESIS

Osmancan ERŞAHİN

Department of Naval Architecture and Marine Engineering

Naval Architecture and Marine Engineering Graduate Programme

JANUARY 2014

ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL OF SCIENCE ENGINEERING AND TECHNOLOGY

PUMPING THE MAIN SAIL FOR PERFORMANCE GAIN

M.Sc. THESIS

Osmancan ERŞAHİN
(508101017)

Department of Naval Architecture and Marine Engineering

Naval Architecture and Marine Engineering Graduate Programme

Thesis Advisor: Prof. Dr. Mustafa İNSEL

JANUARY 2014

<u>İSTANBUL TEKNİK ÜNİVERSİTESİ ★ FEN BİLİMLERİ ENSTİTÜSÜ</u>

PERFORMANS KAZANCI İÇİN ANA YELKEN POMPALAMAK

YÜKSEK LİSANS TEZİ

Osmancan ERŞAHİN
(508101017)

Gemi İnşaatı ve Gemi Makinaları Mühendisliği Anabilim Dalı

Gemi İnşaatı ve Gemi Makinaları Mühendisliği Programı

Tez Danışmanı: Prof. Dr. Mustafa İNSEL

OCAK 2014

Osmancan Erşahin, a **M.Sc.** student of ITU **Graduate School of Science Engineering and Technology** student ID **508101017**, successfully defended the **thesis** entitled "**PUMPING MAIN SAIL FOR PERFORMANCE GAIN**", which he prepared after fulfilling the requirements specified in the associated legislations, before the jury whose signatures are below.

Thesis Advisor :	**Prof. Dr. Mustafa İNSEL** İstanbul Technical University	....
Jury Members :	**Ass. Prof. Dr. Şebnem Helvacıoğlu** İstanbul Technical University	....
	Ass. Prof. Dr. Uğur Buğra Çelebi Yıldız Technical University	....

Date of Submission : 16 December 2013
Date of Defense : 20 January 2014

FOREWORD

I would like to thank my supervisor Prof. Dr. Mustafa İnsel and my parents and my sister for their continued support; Dr. Ahmet Ziya Saydam for his contribution.
I would also like to thank the following people for their valuable contributions:

- Mr. Levent Özgen (Veteran Turkish National Sailor)
- My executives at Tezmarin Tur. ve Tic. A.Ş. notably Mr. Vedat Tezman
- Mr. Özgür Numan (Naval Architect and Licensed Sailor)

December 2013 Osmancan ERŞAHİN
(Naval Architect)

TABLE OF CONTENTS

Page

ABBREVIATIONS

AR_E : Effective Aspect Ratio
c : Chord length
C_D : Drag Coefficient
C_{Di} : Induced Drag Coefficient
C_{Du} : Drag Coefficient not Corrected for Wake Blockage
CFD : Computational Fluid Dynamics
C_L : Lift Coefficient
C_P : Pressure Coefficient
CPU : Central Processor Unit
D : Drag Force
f : Frequency
FEM : Finite Element Method
g : Gravitational Acceleration
h : Static Head
ISAF : International Sailing Federation
k : Reduced Frequency
L : Lift Force
NACA : National Advisory Committee for Aeronautics
P : Pressure
q : Dynamic Head
RNG : Re-Normalisation Group
S : Surface Area
SST : Shear Stress Transport
V : Velocity
Y+ : Wall Distance Unit
α : Angle of Attack
β : Apparent Wind Angle
ρ : Density

LIST OF TABLES

Page

LIST OF FIGURES

PERFORMANCE GAIN DURING PUMPING THE MAIN SAIL

SUMMARY

On this project, it is tried to understand the nature of the flow around the sail on special condition. The writer has done a similar project previously. On this study, it is aimed to understand why sailors are pumping the mainsail on downwind sailing and how much is the performance gain with the help of this motion. Since, pumping the main sail is a hard thing to do and there are lots of constraints to do that. According to sailing rules there has to be a wave coming from the back of the boat and sail can be pumped three times in a row at most. These were most exciting and motivating part of this topic.

Firstly, an extensive literature search is done about the flapping foils. Because no study has been found on this topic, the most familiar ones has been ivestigated. Flapping foil studies are very wide and the nature of the flapping movement is totally known. There are different types of studies done on this topic; some of them are trying to understand the movement of fish tale, some of them are trying to understand the movement of bird wings and some of them are trying to developing a flying vehicle or marine vehicle using flapping foils instead of conventional systems. After these papers were read the air flow behaviours around the flapping foils were understood.

Secondly, a study which includes validation data has been found. The validation data is needed to set up a CFD (Computational Fluid Dynamics) model on CFX. So, it could also work on the main study by extending this model which has been approved against the validation data.

After determining a study as a guide which has both experimental and numerical results on the same study, a model on CFX has been generated. The data was about a pitching foil (NACA 0012) with low angle (2 degrees). The model has been created and the results are compared to the validation data. The results were showing the thrust coefficients on different reduced frequencies. The results were reasonable, so it has been decided to continue with this model.

Following, the tested model modified for the main study. The forces on the main sail were calculated on different conditions like below;

- Non-pitching
- Pitching with three different angles 5,10 and 15 respectively
- Pitching frequencies varied as f=1 Hz and f=0.5 Hz for each angle

As a result, it is seen that there is a performance gain which can not be ignored when the thrust values non-pitching and pitching main sail are compared. On the

performance gain, first important parameter is the pitching angle and second parameter is the frequency. Priority of these parameters are same what sailors say according to their experiences in practical sailing. They state that pumping should be done as big as it can be done and then as much as much it can be done.

In conclusion, it is obvious that pumping main sail brings more performance to the sailors. Although it is very hard and requires strength and condition it has to be done on every possibility. Since, pumping will cause to big differences between the competitors especially on one design races.

ANA YELKENE POMPA YAPMANIN PERFORMANSA ETKİSİ

ÖZET

Yelkenler, yelkenli yatların en temel sevk sistemleridir. Bu yüzden çalışma prensiplerinin (yelken etrafındaki akışın) bilinmesi çok önemlidir. Bunun teknenin dizayn sürecinde bilinmesi zaman ve para tasarrufu sağlayacaktır. Ayrıca yelkenlerden daha fazla performans alınacaktır.

Yazar tarafından yapılan ilk çalışmada yat aerodinamiğinin temeli sayılacak bir çalışma yapılmış ve direk ile yelken etrafındaki akış, hesaplamalı akışlanlar dinamiği yöntemi ile incelenmiştir. Bu çalışmada ise daha spesifiki bir konu seçilmiş ve özel şartlar altındaki bir yelkenin etrafındaki akım incelenmiştir. Yelkencilerin rüzgar altı seyrindeyken neden ana yelkeni pompaladıkları ve bu hareketle ne kadar performans kazandıkları sorularına cevap aranmıştır. Pratikte sürekli uygulanan ve verimliliği gözlenen bir hareketin teorik olarak açıklanması amaçlanmıştır. Ana yelken pompalamak zor bir iş olup, birçok kısıtlamaya sahiptir. Bu kısıtlardan biri olan yelken yarış kurallarına göre eğer teknenin arkasından gelen bir dalga varsa her dalga için en fazla 3 defa pompa yapılabilir. Kısıtların olması ve hareket için özel durumların aranması bu konuyu ilgi çekici ve motive edici hale getirmektedir.

Literatür taraması sonucunda bu konuda hiç çalışma yapılmamış olduğu görülmüştür. Bu nedenle benzer bir hareketin incelendiği çalışmalar aranmıştır. Diğer yandan pratikte sıklıkla rastlanan bir uygulamanın teorik incelemesinin yapılmamış olması şaşırtıcıdır. Yelkenin pompa yapması bir foilin kanat çırpma (flapping) ile özdeşleştirilebileceği için literatür taraması bu alanda yoğunlaştırılmıştır. Kanat çırpma hareketi yapan foiller hakkında bir çok makale bulmak mümkündür. Bu konudaki çalışmalar iki ayrı ana başlıkta toplanmışlardır:

- Biyolojik hareketleri (kuşların, böceklerin kanat çırpması, balıkların kuyruk hareketleri vb.) inceleyenler
- Biyolojik hareketlerden esinlenerek mekanik itme/manevra sistemleri geliştirmeye çalışanlar.

Ayrıca araştırma sonucunda görülmüştür ki bir kanat çırpma hareketi üç farklı hareketin genel adıdır:

- Baş-kış vurma (pitching)
- Kaldırma (heaving)
- Baş-kıç vurma ve kaldırma (pitching ve heaving)

Bu araştırma sonucunda yelkenin pompa hareketinin sadece baş-kıç vurma hareketi yapan bir foil ile örtüştüğü görülmüştür. Baş-kıç vurma hareketinin doğası öğrenilmiş, harekette akımı etkileyen parametreler belirlenmiş ve akımın karekteri saptanmıştır. İki hareket arasındaki tek fark yelkenin pompa hareketinin, baş-kıç vurma hareketi yapan bir foilin yaptığı hareketin yarısı olmasıdır. Baş-kıç vurma

hareketi yapan bir foil başlangıç noktasından sonra önce – (negatif) yöne gider, sonra geri dönerek başlangıç noktasından geçer ve + (pozitif) yöne doğru ilerler. Sonrasında ise geri dönerek tekrar başlangıç noktasına gelir. Böylelikle tam bir baş-kıç vurma hareketi tamamlanmış olur. İşaret kabulüne göre foil önce + (pozitif) sonra – (negatif) yöne de gidebilir. Yelken ise başlangıç noktasından – (negatif) yöne hareketlendikten sonra başlangıç noktasına döndüğünde + (pozitif) yöne ilerlemez. Buradan sonra tekrar – (negatif) yöne gider ve böylelikle birinci pompa hareketi sonlanmış; ikinci pompa hareketi başlamış olur.

Amaçlanan çalışmanın yapılabilmesi için çözüm yöntemi olarak hesaplamalı akışkanlar dinamiği yöntemi belirlenmiştir. Ancak bilindiği gibi bu yöntemin kullanılabilmesi için kurulacak modelin daha önce yapılmış ve sonuçları ispatlanmış bir çalışma ile karşılaştırılması gerekmektedir. Bunun nedeni kurulan yeni modelin verimliliğinin ve doğruluğunun saptanmasıdır. Literatür taraması sırasında bu konuda yapılmış ve karşılaştırma yapılabilecek verileri bulunan çalışmalar süzgeçten geçirildikten sonra deneysel ve matematiksel hesap sonuçlarına sahip bir karşılaştırma çalışması seçilmiştir. Seçilen çalışmada kanat çırpma hareketi yapan bir foilin farklı frekans değerlerinde sadece baş-kıç vurma hareketi ve sadece kaldırma hareketi yaparken oluşturduğu itme kuvveti katsayısının değerleri mevcuttur. Yalın bir baş-kış vurma hareketine ihtiyaç duyduğumuz için bu çalışma karşılaştırma çalışması olarak belirlenmiştir. Literatürün genelinde baş-kıç vurma hareketi ile kaldırma hareketini beraber yapan foiller üzerine çalışılmıştır.

Karşılaştırma çalışmasındaki foil ve ortam aynı şekilde amaçlanan çalışmanın da yapılacağı hesaplamalı akışkanlar dinamiği programında modellenmiştir. Bu çalışmada piyasada bulunan yazılımlardan biri olan CFX programı kullanılmıştır. Modelleme sırasında karşılaşılan ilk problem (kullanılan program 3 boyutlu çalışma yapmak amacıyla tasarlandığı için) 3 boyut etkisini yok ederek modelin 2 boyut etkileri altında çalışmasını sağlamak olmuştur. Bu problem modellenen ortama küçük bir kalınlık verilerek aşılmıştır. Sonrasında ise bir başka problemle karşılaşılmıştır. Karşılaştırma çalışmasında, NACA 0012 profiline sahip bir kanat önder kenar ucundan kanat uzunluğunun %25'i kadar içeride bir noktayı merkez kabul ederek bu nokta etrafında 2 derecelik bir baş-kıç vurma hareketi yapmaktadır. Bu hareket yeni kurulan modelde birebir aynı olarak tanımlanamamıştır. Baş-kıç vurma hareketi karşılaştırma çalışması ile birebir aynı olacak şekilde tanımlanmamış olsa da bu farklılığın etkilerinin amaçlanan ana çalışmada problem olmayacağına karar verilmiş ve bunun gerekçeleri belirlenmiştir. Ayrıca farklılıktan kaynaklanan etkiler göz önünde bulundurulduğunda sonuçların karşılaştırma makalesindeki veriler ile örtüştüğü görülmüştür. Bunun sonucunda kurulan modelin doğru ve verimli bir şekilde çalıştığı kabul edilmiş ve ana çalışmanın modellenmesine geçilmiştir.

Seçilen yelken formu karşılaştırma çalışmasında kullanılan modele entegre edildikten sonra ortam konusunda gerekli değişiklikler (yelkenin ortam içerisindeki pozisyonu, ortamın büyüküğü, akımın giriş açısı/hızı vb.) yapılmıştır. Sonrasında ise farklı frekanslar ve farklı açılar içeren bir matris için bütün hesaplamalar yapılmıştır. Yelkenin hareketini belirleyen ve hesaplama matrisinde yer bulan farklı değerler şu şekildedir:

- Açı 5 derece, frekans 0,5 Hz
- Açı 5 derece, frekans 1 Hz
- Açı 10 derece, frekans 0,5 Hz

- Açı 10 derece, frekans 1 Hz
- Açı 15 derece, frekans 0,5 Hz
- Açı 15 derece, frekans 1 Hz

Frekans dışında açının da bir parametre olarak kullanılmasının nedeni pratikte bunun da etkisinin olduğunun bilinmesidir. Ayrıca böylelikle hangi parametrenin etkisinin daha çok olduğu görülmüştür. Performans kazancının hiç pompa yapmayan bir yelkene karşı görülebilmesi için hesaplama hareketsiz bir yelken için de gerçekleştirilmiştir.

Hesaplamalar sırasında karşılaşılan bir problem nedeniyle birebir gerçek hayattan uzaklaşılmıştır. Ancak buna rağmen pompa hareketi sonucunda performans kazancının olup olmadığı net olarak belirlenebilmiştir. Yelkenin hareketi tanımlanırken gerçek hayatta olduğu gibi anlık pompa hareketleri tanımlanamamıştır. Bu nedenle pompa yapmayan ve farklı hareket özellerinde sürekli pompa yapan yelkenler modellenmiştir.

Sonuçta, pompa yapan ve yapmayan yelkenlerin ürettiği götürücü kuvvet katsayısı kıyaslandığında ciddi bir performans kazancı göze çarpmaktadır. Hesaba katılan ve çeşitlendirilen her iki parametrenin de (açı ve frekans) artışının performans kazancına neden olduğu görülmüştür. Değiştirilen parametrelerin hangisinin etkisinin daha yüksek olduğu da ayrı bir önem taşımaktadır. Bu nedenle her farklı hareket tanımındaki parametreler buna göre değiştirilmiştir. Yukarıda da görülebileceği gibi tanımlanan hareketin birinde açı 5 derece ve frekans 1 Hz iken diğerinde açı 10 derece ve frekans 0,5 Hz'tir. Böylelikle her iki harekette de aynı süre sonucunda yelken aynı yolu yapmıştır ve aynı enerji harcanmıştır. Ama iki hareketin sonucuna bakıldığında performans kazançları farklıdır. Bu da iki parametrenin sonuca etkisinin farklı olduğunu göstermektedir. Her iki parametrenin performans kazancına etkisi kıyaslandığında ise etki sırası şu şekildedir:

1) Açı
2) Frekans

Performans kazancını belirleyen en önemli etken pompa yapan yelkenin bir salınımının ne kadar uzun sürdüğüdür. Yani burada açının büyüklüğü birinci etkendir. Sonrasında ise salınım hızı (frekans) önem taşır. Yelken yapanların deneyimlerine göre parametrelerin önceliği sonuçlarmız ile aynıdır. Yelkenciler pompalamanın öncelikle yapılabildiği kadar uzun (büyük açılı) ve sonrasında ise yapılabildiği kadar hızlı (yüksek frekansta) yapılması gerektiğini söylemektedirler. Teorik olarak hesaplanan sonuçların pratik hayatla örtüşmesi yapılan kabullerin ve hesapların doğruluğunu da ispatlamaktadır.

Kısacası, ana yelkende pompa yapmak yelken performansını arttırmaktadır. Ana yelkeni pompalamak zor olup, çok güç gerektirmesine rağmen, uygulanabilecek her durumda yapılmalıdır. Pompa yapmak fiziksel yeterliliğin sağlanabildiği her tip teknede performans kazancına neden olurken özellikle tek tip tekneler yarışlarında rakipler arasında büyük farklar oluşmasına neden olmaktadır.

1. INTRODUCTION

The sails are main propulsion system of the yachts. That's why it is really important to know the characteristic of the flow around sails and the forces produced by the sails. If this information is known in the design stage, lots of financial resources and time could be saved. And also a better performance from the yacht could be achieved. There are three currently different techniques to analyze the flow around the sails; full scale measurements, wind tunnel experiments and CFD (Computational Fluid Dynamics) calculations. Full scale measurements are not so easy to conduct and also not so efficient because of uncontrolled environment such as the wind direction and speed changes. Wind tunnel experiments have problems while results data transferring to full scale and also it takes more time than CFD on the preparation of the model. In order to use CFD analysis method, characteristics of the sails like chord length, camber ratio and wind velocity, angle of incidence have to be known. Actually, the whole situation with all parameters has to be known to get reliable results. The parameters about the sails are scaled down from the real yachts or theoretically chosen. It means different profiles of sails and mast shapes (elliptical, circular, etc.) could be chosen in order to see the effects. Wind specifications are decided by the researchers according to sailing area of the yachts and sailing conditions of the yachts like downwind or upwind. However, deformations of the sails; form changes when the wind push the sails and changes with the wind; wind has never constant speed or angle variation of which cannot be considered on the CFD analyzes.

After investigating the air flow around the sail on my previous project I have decided to enlarge it on a special topic; pumping the main sail for performance gain. CFD has been used to analyze the wind flow around the sails while pumping on this project. As a starting point for CFD calculations wind tunnel results are needed. However, after getting the first results with CFD which are succesfull compared to experiment results, it is easier to vary the analysis and get results for different conditions of problem.

Pumping the main sail during downwind sailing is a routine technique for the sailors but has not been investigated sufficiently up to now. There isn't any research about the performance gain with this technique. Pumping the mainsail is similar to pitching motion of an airfoil/hydrofoil. Although the frequency, amplitude and conditions are different, it has similar physical properties to the wing flapping of a bird or tail flapping of a fish in the nature.

In order to model sail pumping, it was needed to establish a validation database. Observations of the movement of the animals in nature or making some experiments can be considered to establish a validation database. However, it will take too much time and resources. That's why it is decided to choose a previous research as a validation case for preparing the model of the problem.

At the start of this study the target was modelling a mainsail movement during pumping on downwind. Therefore, the literature research is done to find studies having similar movements. There are different types of flapping wing movements on literature; pitching, heaving, combined pitching and heaving. Pumping the mainsail is similar to pitching motion of a foil. A study has been found in literature and chosen as validation study which has force coefficients while pitching on different reduced frequencies.

In conclusion, a model has been developed to calculate the validation data. After seeing that the results of validation study and the model are matching, the model has been designed for the main problem. According to the results of this study, mainsail pumping has an important positive effect on the performance of the boat on downwind sailing. Although the results are reliable, it would be helpful that experiments are done to be sure about the validity of the results.

2. THEORETICAL BACKGROUND

2.1 Forces Acting on a Yachts

A sailing yacht operates at the boundary where air and water intersect; the sails, rig and the above water part of the hull are placed in air and the underwater part of the hull is immersed in water. A yacht sailing at a constant speed in calm water experiences two forces due to the existence of the two fluids; the aerodynamic force is due to the airflow around the sails and the hydrodynamic force is due to motion in water. In calm water at constant speed, these forces balance, they are equal in magnitude and opposite in direction [1].

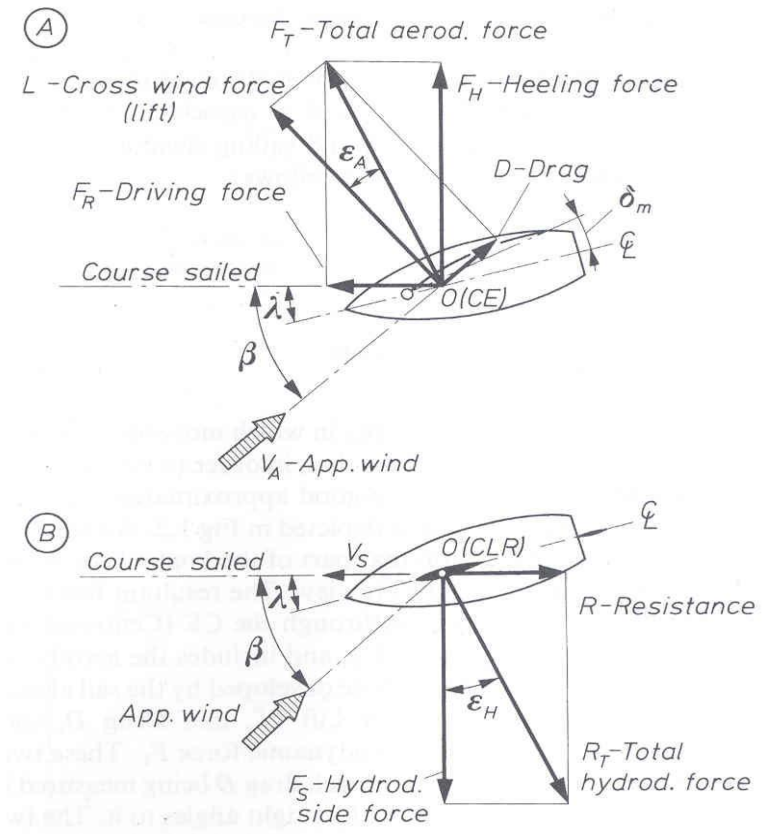

Figure 2.1 : Forces produced by sails [2].

The sails experience the apparent wind, which is the vectorial sum of the true wind and the yacht's speed. Figure 2.1 shows that the total aerodynamic force is consisted of the lift which is perpendicular in direction to the apparent wind and the drag which acts at the same direction as apparent wind [2]. This axis system is named as wind axes. For convenience, by the appropriate translation these forces can also be expressed as driving force, which is acting on the sailed course and heeling force which is perpendicular to driving force and on the leeward side [3].

There are two main components of the hydrodynamic force acting on the hull which are the resistance on the opposite direction of driving force and the side force on the opposite direction of heeling force. These forces, as seen on Figure 2.1 act along the space axis [1].

In order for the hull to produce side force, it has to have an angle of attack to the flow which is named the leeway angle. While the hull is producing side force, the angle between the heading of the yacht (in line with the body axis) and the centre line of the yacht is equal to the leeway angle.

2.2 Aerodynamics of Sails

2.2.1 Flow around sails

The working principle of the sails on the upwind condition is very similar to the aircraft wings. The aim is here producing the required lift and creating minimum drag in order to make the lift/drag ratio maximum.

2.2.1.1 Bernoulli's equation

Sails operate immersed in air; due to the oncoming air flow, pressure acts along the sails on both sides. In order to understand better the physics of the flow around wing sections, Bernoulli's equation is a useful tool. Bernoulli states that the total pressure around a streamline is constant under the assumptions that the fluid is inviscid and incompressible [4].

$$P + \tfrac{1}{2}\rho V^2 + \rho g h = \text{constant} \qquad \textbf{(2.1)}$$

When the approaching flow encounters a wing, a stagnation point occurs near the leading edge that divides the flow into two parts. If the flow is in line with the wing, the pressure distributions along the windward and leeward sides of the wing are the same. When there is an angle of attack, the flow in the windward and leeward sides become asymmetric. The flow on the leeward side of the wing has to move along a longer way due to the angle of attack present. This results in speeding up of the flow on the leeward side. Due to Bernoulli's principle, this accelerated air results in lower pressure than the slower windward side. This small pressure difference becomes measurable when integrated along the whole wing area and it is the main source of lift production. A convenient way to express the pressure difference is the usage of the Pressure Coefficient, Cp [4].

$$Cp=1-(q/v)^2 \quad \textbf{(2.2)}$$

Although the working principles of a wing and a sail similar, we cannot explain the nature of produced lift on a sail on this way. Since the sail has no thickness, we cannot say that the air has to move a longer way on the leeward side. Therefore we have to look to circulation theory of lift.

2.2.1.2. Circulation theory of lift

The popular explanation for the creation of lift on the sails is the circulation theory. This theory was first suggested by Lancaster and then developed by Kutta, Joukowski and Prandtl[2].

If the flow past a highly cambered asymmetrical aerofoil at zero angle of attack is considered, the flow will be divided into two fluid particles; moving towards upper and lower surfaces of the foil section. The point that the flow separate into two is called the stagnation point S1 on Figure 2.2-A.

The two flow parts travel along at equal speeds and since the upper surface is longer than the lower one the particle moving along the lower surface arrives to the trailing edge before the other. Naturally, its tendency is to go around the sharp turn at the trailing edge in order to be combined with the other one as seen on Figure 2.2-B. At the initial stage of the flow, when the two mentioned parts of the flow meet somewhere on the upper surface near the trailing edge a second stagnation point forms, named S2. As the flow develops, due to viscosity of the fluid and strong inertia forces, this state of flow might not be maintained for long. The flow breaks away from the trailing edge forming the starting vortex seen in Figure 2.2-C [2].

As the starting vortex rotates, due to Newton's third law of motion a counter rotation develops around the aerofoil in the opposite direction. This induced counter rotation appears as the circulation around the foil. Due to Newton's third law, angular momentum can not be created in a system without reaction. Quoting reference 1: "All forces arise from the mutual interaction of particles and in every such interaction the force exerted on the one particle by the second is equal and opposite to the force exerted by the second on the first".

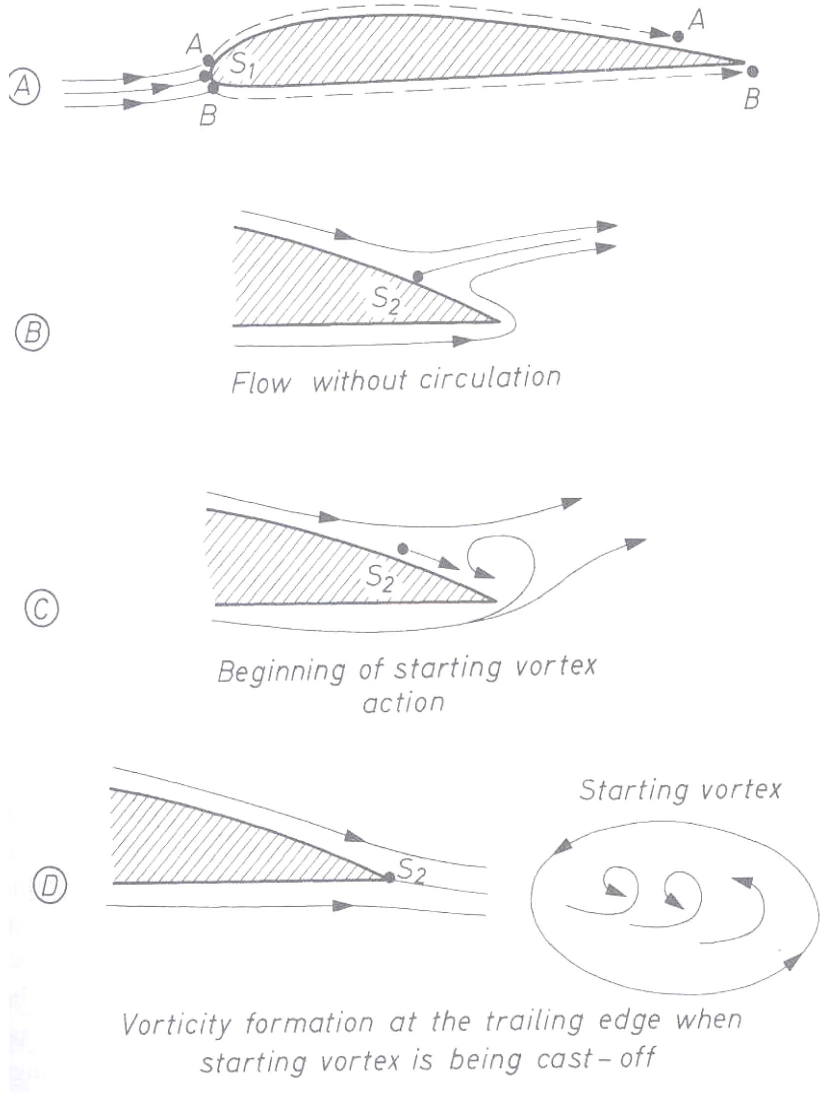

Figure 2.2: Flow around foil [2].

Kutta and Joukowski have stated that as the starting vortex initiates the flow, it breaks away from the trailing edge and moves downstream in the wake. During this state, the stagnation point moves well aft, near the trailing edge in which case there is no difference between the velocities on the upper and lower surfaces while leaving the foil surface. In other words there is no physical implication that the starting vortex will be able to be maintained. The flow reaches a steady state after this point with a constant amount of circulation and hence lift force. The strength of the vortex that has been shed into the wake known as the trailing vortex is dependent on the circulation around the foil. This leads to the fact that if the circulation around the foil is known, the lift force might be calculated [1].

Glauert has stated that due to the presence of negative and positive pressures on the faces of a foil, a spanwise flow will exist around the ends of the foil. The flow is redirected outwards on the pressure side and inwards on the suction side which induces a swirling motion at the trailing edge, dominantly on the tips of the foil. This swirling motion subsequently develops into the vortex sheet named as the trailing vortex which is shed into the wake [2].

Munk has introduced the concept of "field of induced velocities" to aerodynamics, stating that the direction of the streamlines are altered ahead the foil and behind the trailing edge by the tip vortices giving rise to upwash and downwash. This concept will lead to definition of induced drag which will be discussed in section 2.2.3 [2].

The horse-shoe vortex system consists of the starting vortex, the trailing vortices and the bound vortex which are all linked with each other due to Helmholtz Theorem which states that "a vortex once generated can not terminate in the fluid; it must end at a wall or form a closed loop" [5].

2.2.2 The boundary layer

The boundary layer around a body starts growing from zero thickness at the leading edge of the body and as the flow proceeds downstream shear stresses develop at the proximity of the surface due to large velocities in the mainstream and no-slip condition at the surface. This shear stress slows down the fluid particles which are close to the body. These relatively slow moving particles affect the latter ones and slow them down. As the fluid particles move downstream, this action of slowing down spreads further away from the body surface and the boundary layer of slowed down fluids increases its thickness. The usual convention about the thickness of the boundary layer is that it extends to the point where the velocity of that point is 99% of the free stream velocity in the normal direction to the body [4].

Due to viscous effects present in the boundary layer, the forces acting on the body will mainly depend on the way that the boundary layer develops. Near the leading edge, the flow is likely to proceed in a smooth, ordered, streamlined fashion which is named to be laminar flow. As the fluid particles proceed along the surface, the flow develops into an irregular flow consisting of small scale eddies. In this region, the velocities are variable in magnitude and direction. This type of flow is named turbulent flow and it still follows the surface of the body. The change from laminar to turbulent flow is called transition. The pressure distribution influences the point of transition; a favourable pressure gradient (dropping pressure) delays the transition whereas an adverse pressure gradient (increasing pressure) nearly leads to an immediate transition. In case the magnitude of the adverse pressure gradient is adequate enough, it might lead to a completely different case where the flow will not be able to maintain contact with the body surface; leading to separated flow. Near the leading edge, a high adverse pressure gradient is required to cause separation and a favourable pressure gradient can result in a reattachment of the flow. In this case, a leading edge separation bubble forms [3].

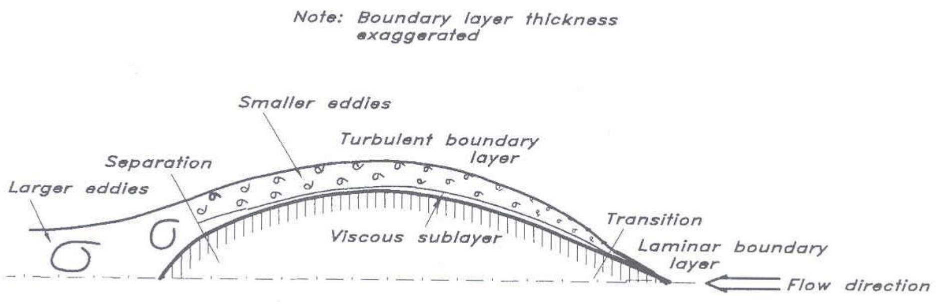

Figure 2.3: Boundary layer around foil [6].

Figure 2.3 visualises the concept of laminar flow, transition, turbulent flow and separation as well as shows how large eddies form downstream of the body and form the wake region [6].

2.2.3 Drag components of sails

Figure 2.4 shows how to split the drag of the sails into components. This sketch does not include the windage drag caused by the existence of the rigging, superstructure and the non immersed portion of the hull. The profile drag includes friction drag and pressure drag (viscous drag) which mainly depend on the Reynolds number and the geometry of the sails (camber, thickness, etc…). The remaining part is the induced drag which is generated by the formation of lift [2].

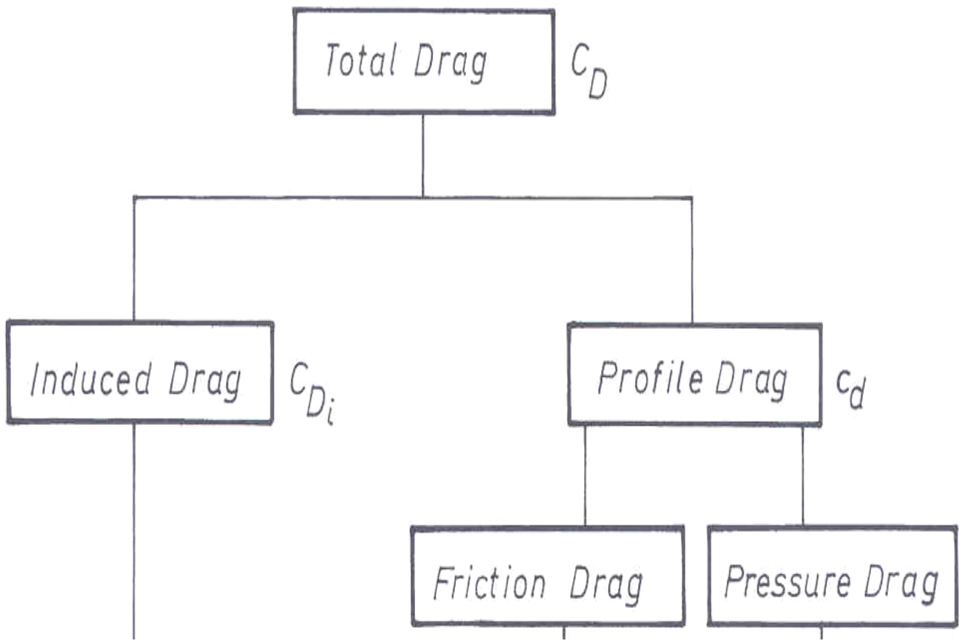

Figure 2.4: Drag breakdown [2].

The induced drag depends on many factors; mainly the span wise distribution of lift, aspect ratio, taper ratio and the amount of twist. They all depend on the trim of the sails. In terms of lift distribution, an elliptical lift distribution leads to the least amount of induced drag. In order to achieve an elliptical lift distribution the sail shape does not necessarily has to be elliptic, since the loading depends on section shape as well as angle of attack and chord wise geometry [2].

$$C_{Di} = C_L^2/(\pi \times AR_E) \qquad \textbf{(2.3)}$$

The above equation is a representation of the induced drag; it varies with the square of lift coefficient and the inverse of aspect ratio. In order to identify the induced drag of sails, a useful way is to plot the drag coefficient versus the square of the lift coefficient as seen on Figure 2.5 [2]. Apart from the region of separated flow, this curve is linear and the slope of this plot is determines the aspect ratio which is fundamentally based on the effective rig height. Therefore, it can be assured that the most important parameter acting on the generation of induced drag is the effective rig height.

2.3 Sailing Conditions

2.3.1 Upwind

The most important parameter in upwind sailing is the ratio of lift to drag. On this sail condition the lift is not alone important because the lift has a huge effect on the heeling force that is not wanted. The lift to drag ratio is the tangent of the drag angle which is the angle between the lift and the total force. A large drag angle implies a low lift/drag ratio.

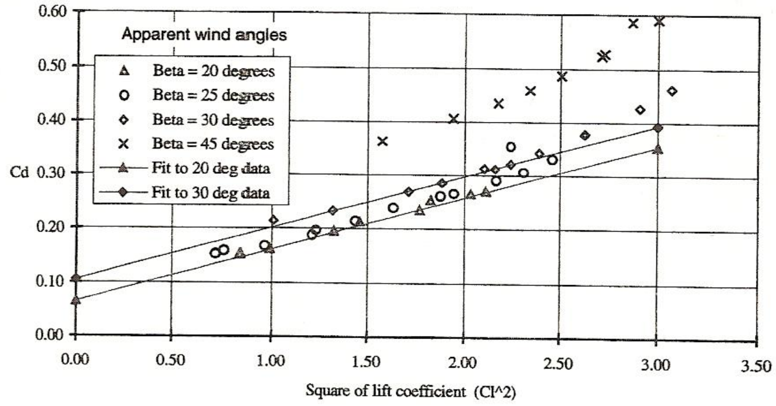

Figure 2.5: C_d against C_l^2 [3].

The use of the drag angle for the aerodynamic and hydrodynamic forces leads to the derivation of the "Beta Theorem". According to this theorem the angle between the apparent wind and the course sailed is equal to the sum of the sail and hull drag angles. Naturally, both in aerodynamics and hydrodynamics maximum lift/drag ratios lead to a better upwind performance [7].

2.3.2 Downwind

The sail performances on the downwind can not be evaluated perfectly as upwind although the nature of the downwind sail aerodynamics is known. Since, the sails have sharp (high angle) leading and trailing edges. It causes to too much separation which affects local pressure distribution and damage the boundary layer on leeward of the sails or even not to be formed. Briefly, estimation of the downwind performances of the sails is hard because places and sizes of the separations are not known certainly.

In downwind sailing, the aim is to produce the maximum drive force as upwind sailing. However, there isn't only one way to reach maximum drive force. Different conditions and aims for downwind sailing are shown on Figure 2.6.

- On beam reaching; the sails are mainly lift producing and it is the main contribution on the drive force. The aim is to maximize the lift.

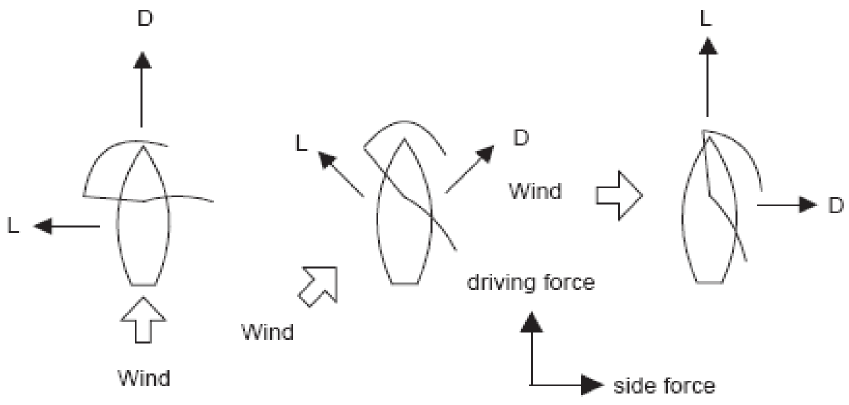

Figure 2.6: Lift and drag forces on downwind [8].

- On broad reaching; both lift and mainly drag forces contribute to the total drive force. The aim is to maximizing lift and drag together.
- On running; the driving force caused mainly by drag [8].

2.4 Flapping Wing Aerodynamics

There are several studies done about flapping wings. Some of them are to understand movement of a fish or flying a bird/insect. And the rest are about creating new vehicles using the same method of a fish tale or a bird/insect wing[9].

We can subclassify flapping wing in two different ways: one is the continuity of the movement and the other one is the type of movement. The continuity of the movement differs as starting/manoeuvring(transient) and propulsion(periodic) [9]. And the types of the movement are heaving, pitching or heaving and pitching, which can be seen on Figure 2.7.

The propulsive force strongly depends on the details of the kinematics of the foil[9]. The kinematic values of a flapping wing are geometry of the wing, amplitude of heaving/pitching, frequency of heaving/pitching and phase shift between the heaving and pitching motions.

Flapping foils produces thrust by creating vortices [11]. Flapping wings produce drag vortices on the low amplitudes like Von Karman street. However, on the high amplitudes (bigger than 5) the vortices have reverse direction which leads to more thrust [11].

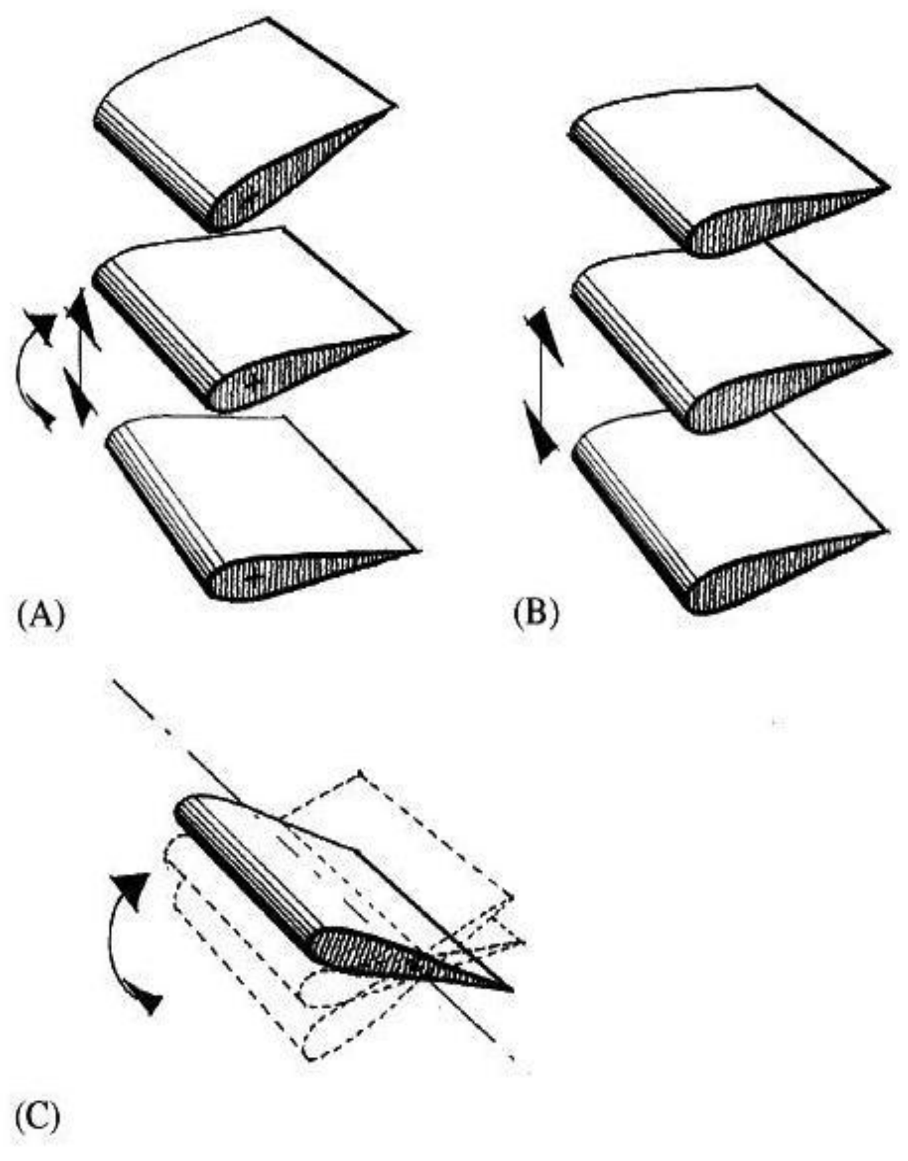

Figure 2.7: Different flapping foil movements [10].

Same effect is also seen while growing of frequency but the transitions are not as clear as amplitude increasing. That's why there is no value called as critical frequency [11].

According to Guglielmini and Blondeaux [9] the positive points of the propulsive flapping fins compared to a conventional thrust system,

1- Combining the function of propulsor, control device and stabilizer thus providing a high manoeuvrability
2- Possessing a sufficiently high efficiency.
3- Having relatively low aerodynamic drag in the "switched-off" position.
4- Being a relatively low-frequency system and having less mechanical problems
5- Operating efficiently in different regimes of motion.

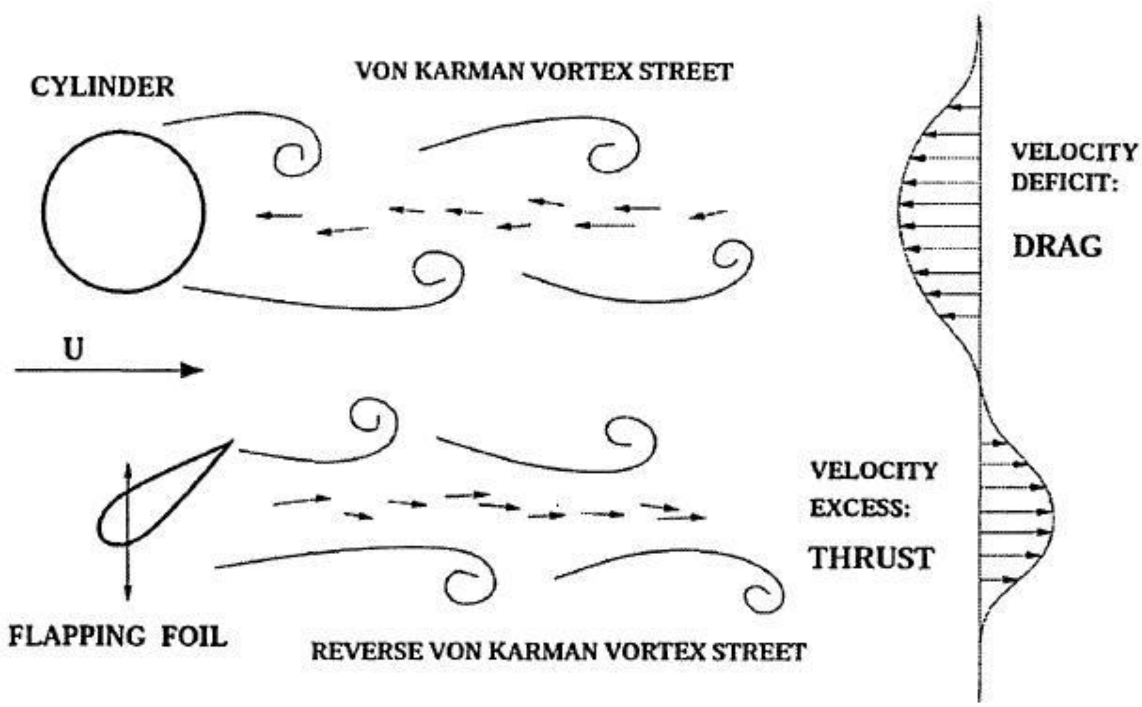

Figure 2.8: Vortices behind the cylinder and flapping foil [11].

3. NUMERICAL MODELLING

Although numerical flow modelling was known for a long time, it could not be used inclusively because the existing computers were not capable [12]. After the technology improvements, the computers start to be powerful and numerical modelling became a possible problem solving system.

Flapping foils have been studied both by numerical modelling or experiments. Both numerical modelling and experiments results found in the literature have been used for validation in this study.

3.1. CFD

CFD is utilised to generate flow simulations through computers in order to solve problems related to fluids by using numerical methods and algorithms. Especially on the complex problems, CFD is utilised to solve the set of partial differential equations. CFD is some kind of FEM (Finite Element Method). The domain must be split into large number of small cells, which are called mesh or grid. And all governing equations are solved for each cell in an iterative way.

3.1.1. History of CFD

All of the CFD problems based on Navier-Stokes equations. CFD is used first to solve the linear potential equations. 1930’s it is started analyzing the flow around a wing in 2-D. Until start using computers it is not used for 3-D analysis. By developing computer technology and turbulence model codes CFD became today’s technology [12].

3.1.2. Usage of CFD

CFD is used on the fields below:

- Aero dynamical design of air and land vehicles

- Hydro dynamical design of ships
- Interior and exterior designs of buildings
- Calculations acting on offshore marine structures
- Biomedical engineering (blood flow in veins)
- Modelling the underwater explosions for Navy
- Estimation of meteorological events

With increasing popularity of CFD, lots of software companies involved to CFD market. Some of those programs are PHOENICS, FLUENT, FLOW3D, STAR CCM+ and CFX, from which the last one is used on this project. The most important differences of CFX from other CFD program all over the world are;

- combining an advanced solver with powerful pre- and post-processing capabilities
- an advanced coupled solver which is both reliable and robust
- an intuitive and interactive setup process, using menus and advanced graphics.

3.1.3. Why CFD?

Advantages of CFD usage:

- Decreasing the expenditure on the design process
- Creating the models and simulations which cannot be analysed experimentally
- Providing detailed information about the analyzed problem

On the other hand, there are also some unfavourable points for CFD. CFD can not be used directly in order to analyze the problems involving turbulent flow. Since, it is not easy to model the whole nature of the problem. There are several turbulent models in order to reach to reality. However, experience is required in order to choose one or as usual the results should be compared with traditional methods for the validation of methodology. There are some universities and companies using their own turbulence codes. They have written these codes after lots of experiments and analysis.

CFD results should be compared with towing tank or cavitation tunnel results for hydrodynamic problems; with wind tunnel results for aerodynamic results. After comparing some experiments it is possible to choose the turbulence model. After validation of the turbulence model the experiments can be left and the rest of your analysis can be done by CFD. Then the best part of CFD will start, time and money can be saved.

The advantages of CFD which make it popular are mentioned on the reference 9:

- CFD allows numerical simulation of fluid flows, results for which are available for study even after the analysis is over. This is a big advantage over, say, wind tunnel testing where analysts have a shorter duration to perform flow measurements.
- CFD allows observation of flow properties without disturbing the flow itself, which is not always possible with conventional measuring instruments.
- CFD allows observation of flow properties at locations which may not be accessible by (or harmful for) measuring instruments. For example, inside a combustion chamber, or between turbine blades.
- CFD can be used as a qualitative tool for discarding (or narrowing down the choices between), various designs. Designers and analysts can study prototypes numerically, and then test by experimentation only those which show promise.

The biggest problem of CFD is that it is not efficiently usable on the personal computers. Some super computers are needed especially if a 3-D model with intensive meshes and complex turbulence models is employed. And also extensive real time is needed for the calculations. Therefore CFD is not so common on the industry and mostly used by researchers.

3.1.4. Analysis process on CFD

3.1.4.1. Preprocessing

The CFD model can be developed bu using the following steps;

- Defining the model; the geometry and control domain,
- Meshing the model; the volume will be divided into small cells, uniform or non-uniform,
- Defining the physical model; equations of motion, radiation, etc. ,
- Defining the boundary conditions, specifications(like velocity, pressure, etc.) of initial conditions of fluid.

3.1.4.2. Processor

As mentioned above, using FEM predict the unknown flow parameters and solve the equations on the turbulence model with this data. It continues iteratively until the errors become under the limit which is defined by the user.

3.1.4.3. Post-processor

Finally a post-processor is used for the analysis and visualization of the resulting solution because evaluating the graphics is easier.

4. CALCULATION WITH CFD

4.1. Validation Database

4.1.1 Preparation of model

As mentioned above, this project is performed by a CFD program called CFX. The data of the model is taken from [10] and the results of this project are also compared with its results. Since NACA 0012 foil were used on the reference 10, the model on this project has been prepared using the same geometry. On the reference 10, the airfoil has been studied for both pitching and heaving combined pitching motion. We have considered just pitching motion for our comparison as sail pumping is concerned with pitching motion only.

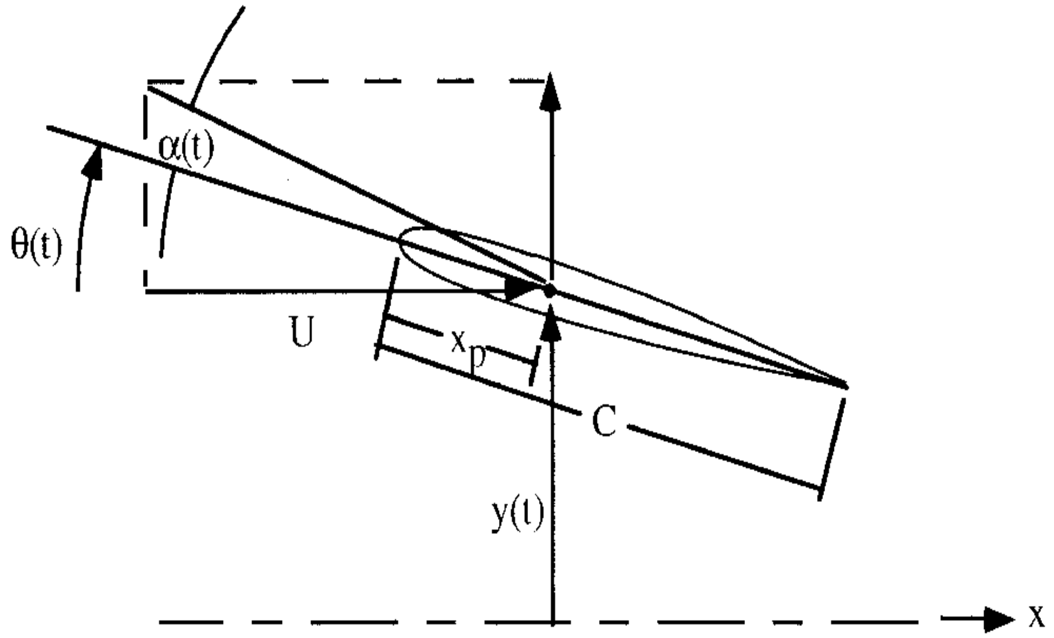

Figure 4.1: Schematic of the pitching and heaving airfoil [13].

The study chosen for validation had a Reynolds number of $1.2x10^4$, pitching angle 2 degrees, distance of origin of pitching motion from the leading edge 0.25 of chord length. The coordinates for the NACA 0012 were taken from the data supplied by Abbott [14].

Because the thrust coefficient (dimensionless value) is going to be compared and there is no value defined on the validation data chord length is chosen as 1 m.

The NACA 0012 profile is drawn on Rhinoceros and exported to CFX. Then the domain is drawn with the CFX interface which is a rectangle 4x8 m decided by the experience. The wing was in the middle vertically and leading edge was in 2m distance of inlet part of the domain horizontally. While using CFX all the models have to have a thickness. It means, they have to be modelled 3-D. The sketch is extruded the model with a thickness of 0.05m in order to consider the model still 2-D. The validation studies are 2-D numerical models which is clear with their mesh cell numbers.

4.1.2. Meshing the model

After importing the model to the mesh interface of CFX, the mesh specifications are defined and the model is meshed. It can be seen on Figure 4.2. There is an inflation (term on CFX using for intensive meshing on boundary layer) with the height of 0.1m in total and consisting of 10 cells. This can be seen on Figure 4.3. And the model has 26.410 cells which is enough for this kind of problem.

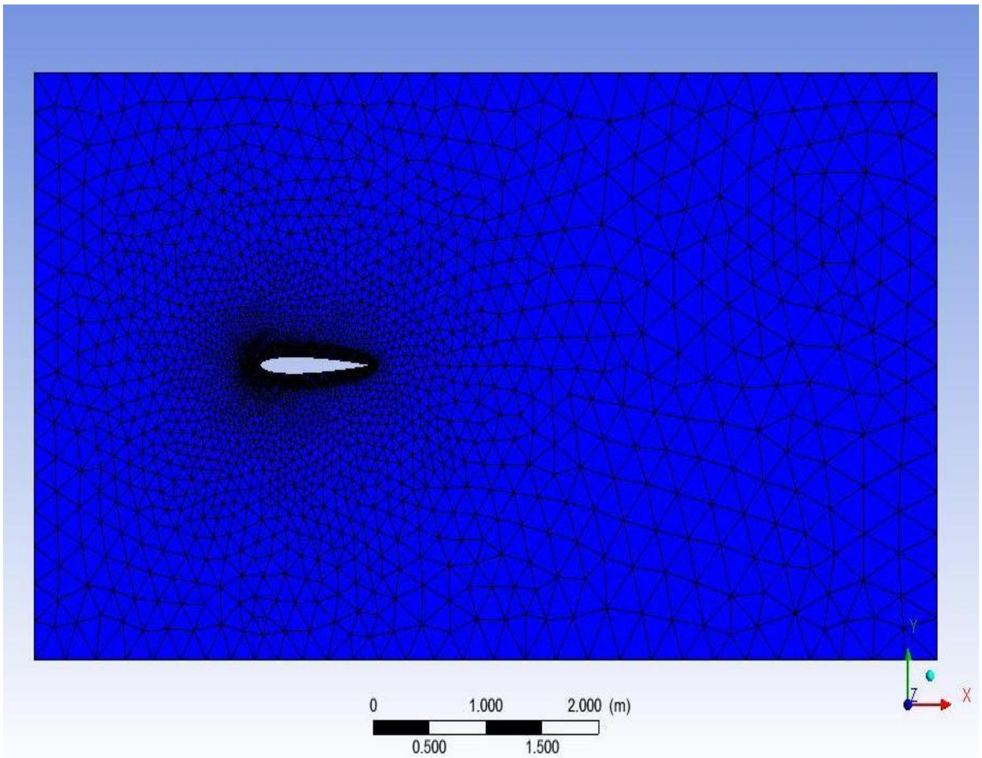

Figure 4.2: Mesh view of whole domain.

4.1.3. Defining the properties of flow and domain

This problem has to behave Two Dimensional. Because there is a movement the problem has to be solved transient instead of steady. Domain fluid chosen as air among the default domains of CFX.

The surface facing the leading edge is defined as inlet having constant speed 10 m/s parallel to the chord. The sides of the control domain and the surface on the back are defined as opening. Last, the top and bottom surfaces are defined as wall in order to keep 2-D conditions on the model.

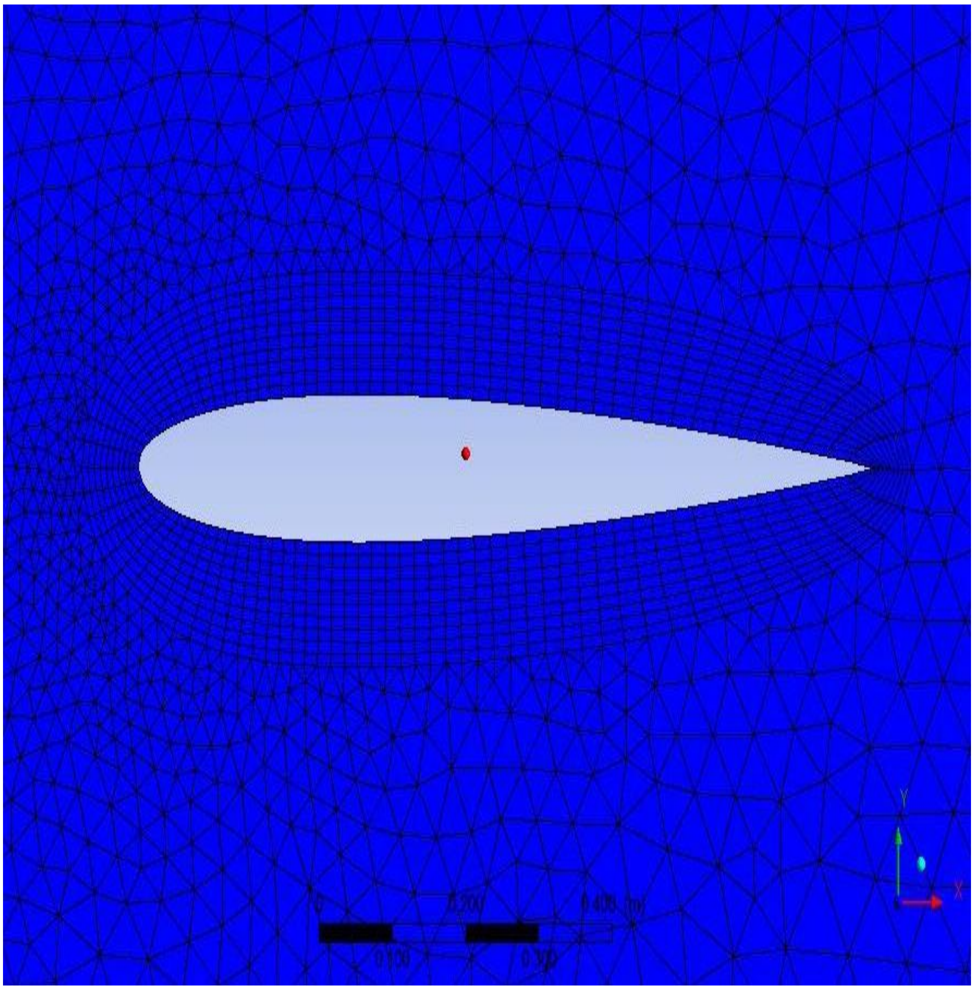

Figure 4.3: Inflation of the mesh on boundary layer.

With the properties of the model and flow the Reynolds Number is around 9 x 10^5. This means the flow is turbulent which is different than the validation data. The reason is to do that is going to be explained on the solution process and results comparison.

4.1.4. Defining the movement of the foil

This was the hardest part of the modelling. There are several solutions on CFX to move the objects such as deforming the meshes or modifying the coordinate system.

All possible systems have been tried to create the movement and pitch the foil. Finally, the rigid body system is used in the current study. A rigid body is defined on the foil and let to move.

A fatal error occured during the calculations which affect the origin point of pitching. It was impossible to choose 0.25c as the centre of the movement because the mesh were collapsing. The centre of the movement had to be placed out of the foil to 0.01c which means almost to the centre of leading edge. The effect of this unavoidable change is going to be discussed on results later.

4.1.5. Choosing the turbulence model and wall treatment

There are different turbulence models; some of them are the same on the all programs and some of them differs from program to program and some of them just used in-house which are written by the universities or commercial companies working on CFD analysis. Commonly used turbulence models on CFD are summarised and their suitability for our problem will be assessed.

4.1.5.1 General terminology of wall treatment

The most important parameter about the wall treatment process is the wall distance unit, named Y+. It is equal to the distance from the wall multiplied by the ratios of frictional velocity and free stream velocity. Y+ is the main control that the user has over wall treatment. Different turbulence models demand different Y+ values for the first cell attached to a wall since different turbulence models are valid down to different regions of the boundary layer owing to the different assumptions associated with them [1].

4.1.5.2 Laminar model

Laminar flow is governed by the unsteady Navier-Stokes equations. Because there is no turbulence model can be used just for laminar flows where Reynolds number is low (<1000) [15].

4.1.5.3 The zero equation model

The Zero Equation model implemented in CFX is simple to implement and use, can produce approximate results very quickly, and provides a good initial guess for simulations using more advanced turbulence models. The model can not be used to obtain final results [15].

4.1.5.4 K-Epsilon model

It has been the most widely used turbulence model for engineering applications. It is a two equation high Reynolds number model which is robust, economical and it has reasonable accuracy for a wide range of flows. It has weaknesses such as flows involving streamline curvature, swirl, rotation, separation and low Reynolds number. Even it is highly popular in the industry, it should be handled with precaution and its suitability for the particular problem has to be assessed carefully [16].

4.1.5.5 RNG K-Epsilon model

RNG K-Epsilon model is an alternative to K-Epsilon model. It is based on renormalization group analysis of the Navier-Stokes equations. The transport equations for turbulence generation and dissipation are same with the K-Epsilon model but the model constants differ [15]

4.1.5.6 Wall treatment for K-Epsilon models

All the K-Epsilon models are valid in the turbulent core region and through the log layer. Therefore, special wall treatment is necessary since the equations cannot be integrated down to the wall. For high Reynolds number flows and flows without complex near wall phenomenon such as strong body forces, severe pressure gradients, rapidly changing fluid properties; standard wall functions might be used where the boundary layer shouldn’t be resolved down to the wall. For adverse pressure gradients and separation effects, non equilibrium wall functions are more suitable. If there is still an uncertainty that using wall functions is not appropriate, the boundary layer might totally be resolved and the option of enhanced wall treatment might be used. These results in an excessive computational effort since the number of cells required to resolve the boundary layer are too high compared to using wall functions [16].

4.1.5.7 K-Omega model

Another two-equation model. Most widely accepted in the aerospace and turbomachinery fields. Accurate and robust for large range of boundary layer flows with pressure gradients, as well as for low Reynolds number flows [17].

This model is a popular alternative for all the K-Omega models. It has a weakness of strong sensitivity to inlet turbulence specifications. Also, the mesh needs to be resolved down to the wall. Therefore, coarse meshes cannot be used with this model. It can take into account transition effects optionally [16].

4.1.5.8 SST model

This model was developed to be used for flows that cannot be accurately predicted with the available K-Epsilon and K-Omega models. introduced a modification to the linear constitutive equation and dubbed the model containing this modification the SST K-Omega model. The SST model has seen fairly wide application in the aerospace industry, where viscous flows are typically well resolved and turbulence models are generally applied throughout the boundary layer [18].

4.1.5.9 Wall treatment for K-Omega models

The only option of wall treatment for K-Omega models is the enhanced wall treatment. Therefore, the boundary layer should be resolved down to the wall [16].

4.1.5.10 Reynolds stresses model

This model is commonly used where the turbulent transport or non-equilibrium effects are important. It naturally includes the effects of streamline curvature, sudden changes in the strain rate, secondary flows or buoyancy compared to turbulence models using eddy-viscosity approximation [15]. Therefore six equations are solved in a three dimensional flow problem. This ends up roughly in a 50% more CPU time and 20% additional memory requirement. Also, the strong coupling between Reynolds stresses and the mean flow results in excessive number of iterations for obtaining convergence. However, it has superior performance in flows involving streamline curvature, swirl and rotation [16].

4.1.5.11 Wall treatment for Reynolds stresses model

The wall treatment for Reynolds stresses model is analogous to K-Omega models where a decision has to be made about the way of resolving the boundary layer [16].

4.1.5.12 General guidelines for turbulence modelling and wall treatment

It is a known fact that there is no universally valid turbulence model that gives

accurate results for all diverse types of flows. As a result of this, the effect of using different turbulence and the sensitivity of the results on this variation should be assessed. If wall functions are being used, the grid should be arranged so that the Y+ values are in the appropriate range specified by the software vendor. Also, it is recommended to avoid triangular or tetrahedral elements on boundary layer grid [19].

4.1.6 Solution process and results

K-Epsilon model is chosen as turbulence model according to the experiences from previous studies. And also it is thought that the specialities below of this model will be useful for this solution:

- having curvature correction algorithm
- having robust wall function algorithm
- enabling a relatively coarse boundary layer resolution

On this problem, there is an airfoil which is pitching with the same frequency constantly. The shape of the domain can be seen on Figure 4.2. The flow is coming to the airfoil with constant speed and airfoil produces forces with pitching motion. CFX can supply the tangential and normal forces produced by the foil on every global axises of the domain. As it is seen on Figure 4.2, X axis is in line with chord of the foil (from leading edge to trailing edge). Y axis is vertical to X axis as usual and the positive part of Y axis is on right side of the foil while foil is facing to the flow. So, the forces produced on X axis are showing the drag or thrust of the foil. They are defining the movement of the foil; forward or backward or no movement. If the force value on X is positive it means drag and vice versa is thrust. The produced forces on Y axis are showing the lift produced by the foil which are not a topic of this study. Moreover, because the foil and motion are symmetric there will not be a net force on Y axis in total.

Then, the calculations are made for different reduced frequencies. The term reduced frequencies was not explained on reference 10. The reference 13 has been observed to get the equation of reduced frequency.

$$k = 2 \times \prod \times f \times c \,/\, (2 \times V) \qquad \textbf{(4.1)}$$

The reduced frequency is an important notion because the validation database is given by different reduced frequencies. In order to get same reduced frequency values with validation database, the current study is calculated for frequencies 15, 20, 25, 35, 47.5 and 62.5 respectively.

After first solution trial, it has seen that the domain size is proper and the Y+ values are good. Expected values of Y+ are between 30 and 300. We have the values between 22 and 412. The values out of the expected range are in a very small area. Besides, wall functions of CFX can use the large values of Y+.

The results of current calculations which can be seen on Table A.1 shows that force produced by flapping foil is changing from drag to thrust with the increase of frequency. This is same with what it is also mentioned on the literature background. Tangential forces on X axis are the frictional part of the drag. It stays same despite of frequency inrease because it depends on the geometry and the geometry does not change. Normal forces on X axis are induced drag and they turn from drag to thrust with the increase of frequency. The flapping foil starts to go further on the reduced frequency values bigger than app. 11.

On the Figure 4.5, the results of current study are compared with the validation database which can be seen on Figure A.7. The pressure contribution can be observed on the appendix. Also, the streamlines are added to the appendix in order to show that the model is working on 2-D conditions.

It is seen that the results of current calculations are slightly higher than the results of the Navier-Stokes on the validation data although they are on the same level. Thrust coefficient grows with flapping frequency [10].

Experimental results are increasing much more faster than both of our calculations and Navier-Stokes validation results. The reason of this discrepancy is the strong leading edge dynamic stall vortices [10].

There is an important difference between calculation processes of the validation data and current study; the origin point of picthing. The pitching point on the validation data is inside of the foil by 0.25c. However, it is out of the foil by 0.01c on current study. In order to see the effects of change on the origin point of pitching, the origin of the pitching has been changed from 0.01c to 0.05c and noticed that the result values of thrust coefficient is going slightly bigger (5%). So, it can be said that the difference caused by the change of the origin of the movement. The reason of this increase is that current study is doing pitching and heaving together compared to the validation data. Because no study is found including helpful data in order to compare picthing foil with pitching and heaving foil, the positive effect of heaving motion can not be known exactly.

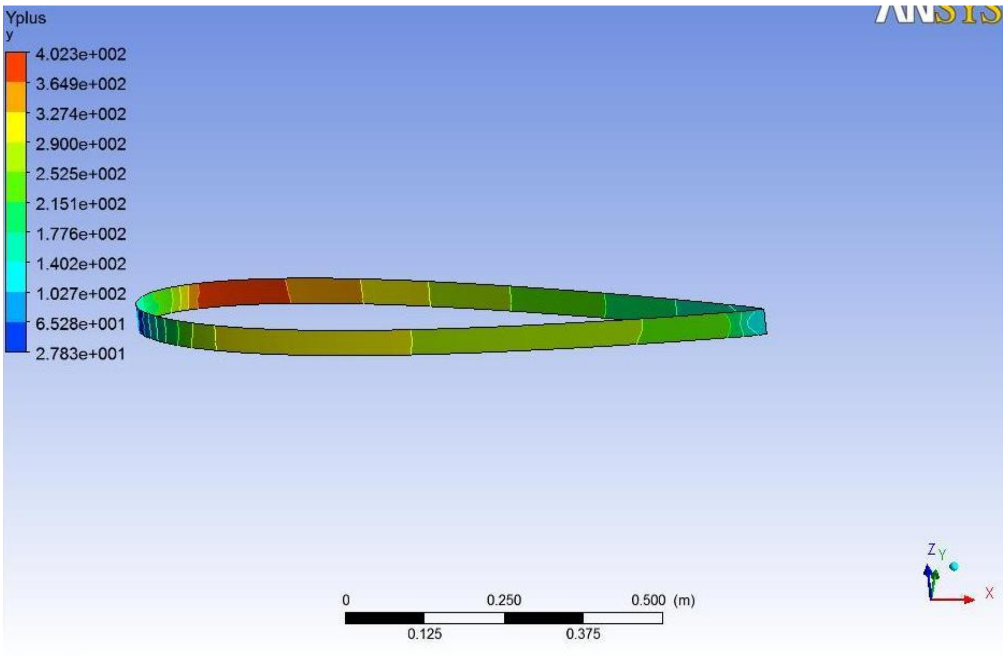

Figure 4.4: Y+ values.

$$C_T = T / (0.5 \text{ x } \rho \text{ x } V^2 \text{ x } S) \quad \textbf{(4.2)}$$

On the other hand, as it is mentioned above, this study is done with high Reynolds number. Since, the main study of this thesis is the pitching(pumping) of a mainsail.

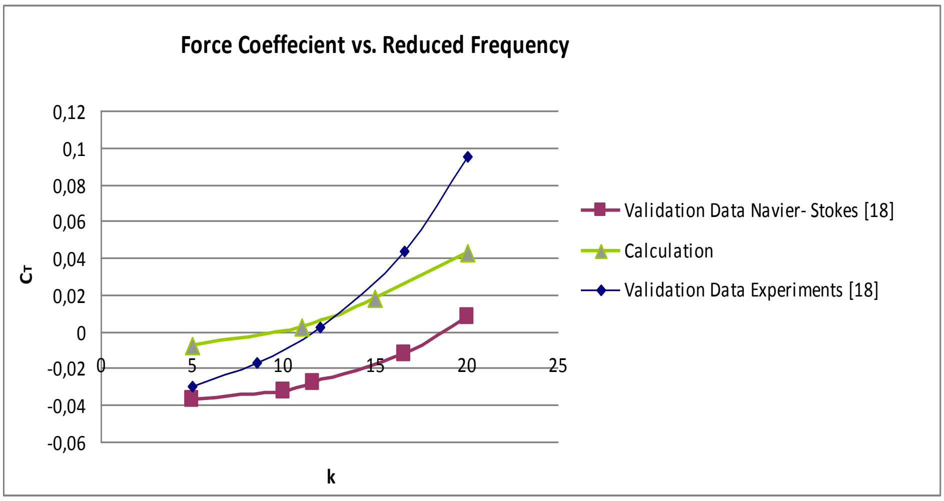

Figure 4.5: Comparison of validation database and current study [10].

We have to know how our model is reacting on the high Reynolds number. Besides, it is mentioned on the theoretical background that flapping foils producing thrust with the help of vortices created by them.

The results of the calculations made with CFX could be considered as reliable conclusions and reasonable enough in order same model to be used for the main study.

4.2. Mainsail Pumping

4.2.1. Defining the problem

The idea of this study is calculating the performance gain while mainsail is being pumped. The sailors are doing that on very large angle broad reach or running. The aim to do that is increasing the boat speed and staying longer time on the waves coming from back and surf with it. Practically, it is seen that there is a performance gain which can not be ignored.

4.2.2 Preparation of model

As we continue on 2-D we are not going to consider the effect of the waves on the boat. First thing in order to prepare this model is having a sail profile. Actually, even a plate can be used as a sail profile for this study which is clear after checking the literature about the downwind sails. However, because this study is one step further work to understand sail aerodynamics a sail profile that has been used by author previously [20].

In order to measure necessary forces for our study easily we have imagined that the boat is staying parallel to the domain. Because there has to be an angle on the inlet flow, the boat is not on the centre. The first sketch of the domain was like on Figure 4.6.

While creating the model, 0.01m thickness is given to the sail because it is needed by CFX to mesh the model. The model of the sail is drawn on Rhinoceros and exported to CFX. The chord length is taken as 1 m. Then the domain is drawn with the CFX interface like on the sketch above. After several trials the domain is obtained to final specifications. Whole domain area is extended step by step to catch the vortices and the wake. The final view of the domain can be seen on Figure 4.7.

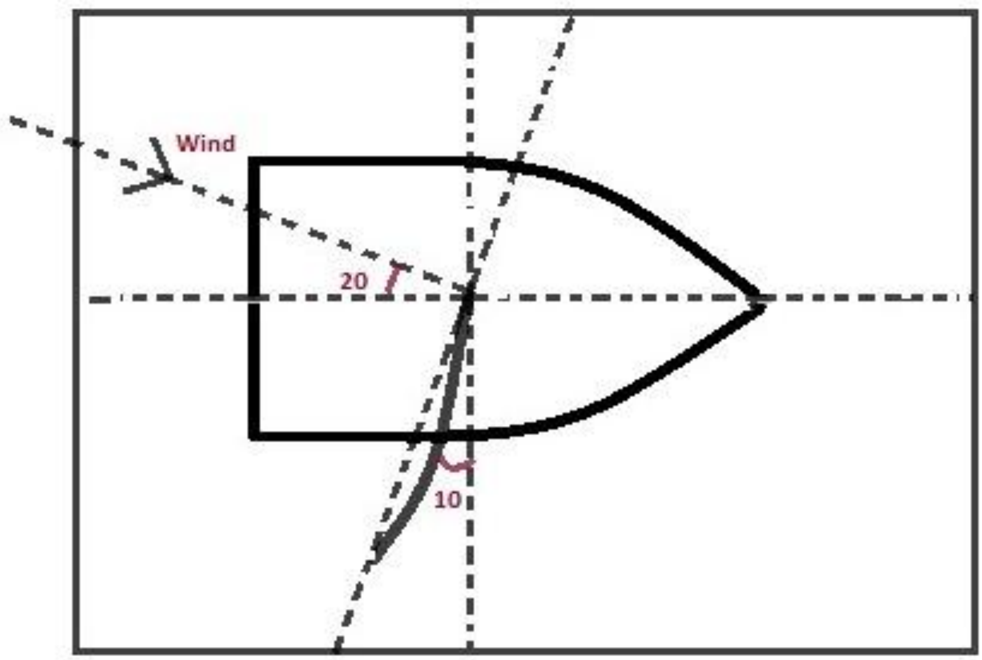

Figure 4.6: Sketch of domain.

As explained above on the modelling of validation study domain all models calculated by CFX have to have a thickness. That’s why this domain has also a thickness of 0.05m in order to consider the model still 2-D same like validation study domain.

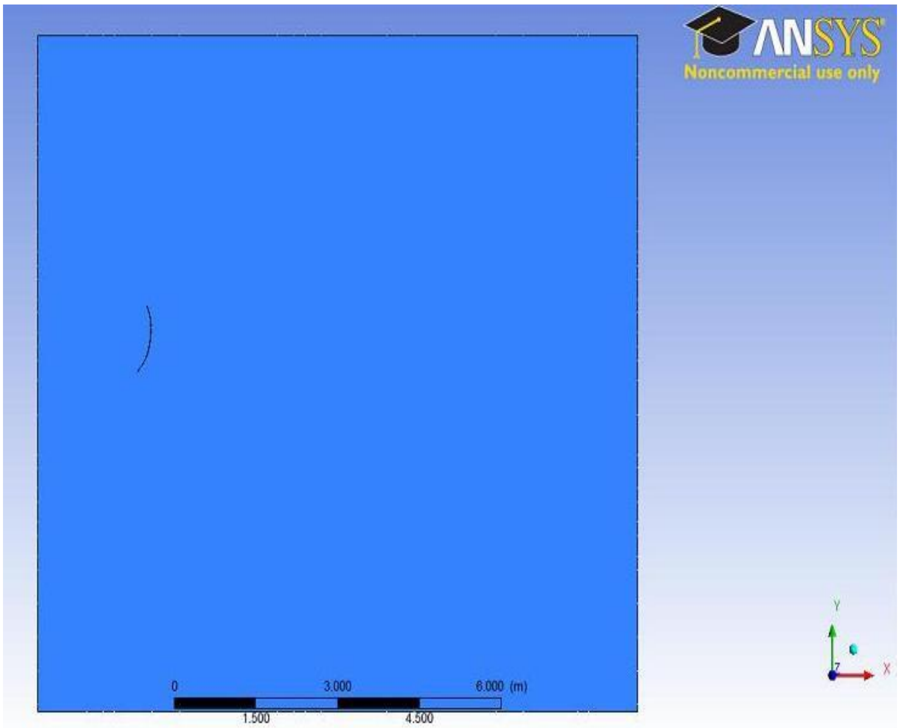

Figure 4.7: Domain of main sail pumping.

4.2.3. Meshing the model

After importing the model to the mesh interface of CFX, the mesh specifications are defined and the model is meshed which can be seen on Figure 4.8. There is an inflation with the height of 0.5m in total and consisting of 30 cells which is shown on Figure 4.9. And the model has 82.204 cells which is much more than validation model. However, there is more and bigger vortices and a bigger domain. That's why it is needed.

4.2.4. Defining the properties of flow and domain

The flow and domain properties are same in general with the properties on validation flow.

The surfaces on sides are defined again as inlet and openings. Top and bottom have been defined as wall since the aim is modelling a 2-D domain. Inlet has constant speed 10 knots with an angle of 20 degrees to the centreline of the domain (can be seen on the sketch above). Because it has thought that a boat having 1 m chord length sail (like a optimist) will have enough power to surf if there are big enough waves and the flow will stay on turbulence stage.

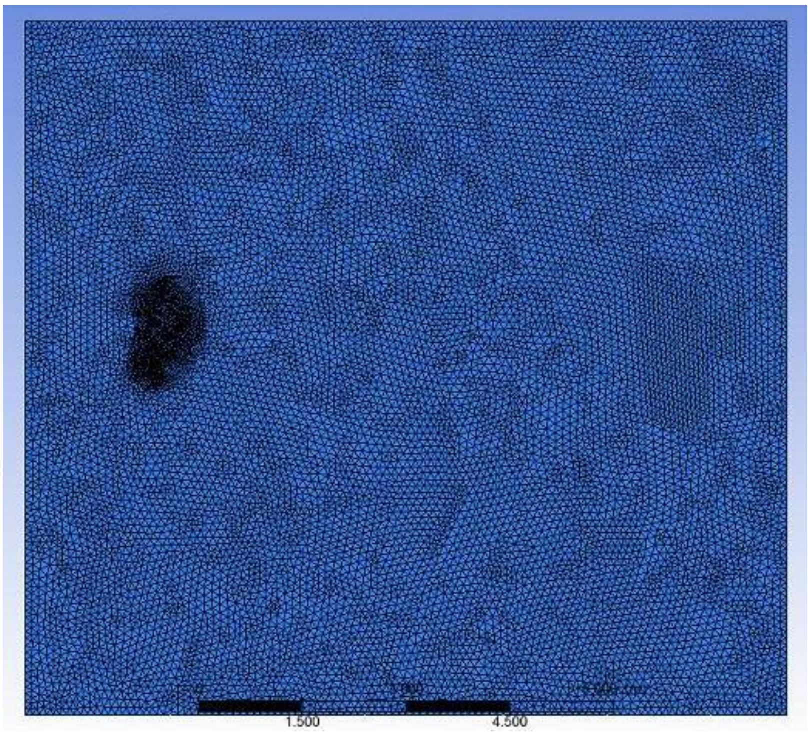

Figure 4.8: Mesh of the main sail pumping domain.

4.2.5. Defining the movement of the foil

The motion model is same with the validation domain. Pumping origin (It is not called as pitching because the pumping movement of the main sail is a half pitching motion) of the main sail has to be chosen as 0.01c out of the leading edge instead of centre of leading edge. The reason is the same mentioned above on the validation model. The results will not be affected on this change because it is too small.

4.2.6 Solution process and results

K-Epsilon model is used as turbulence model after successful results on the validation study.

On this part of the study, there is a sail section which is pumping (semi-pitching) with the same frequency constantly. During one pumping process sail is turning from the leading edge with the given angle and move towards to incoming flow. However, on the way back it is not turning away according to starting line. It stops at the starting line and turning again towards to incoming flow. CFX can supply the tangential and normal forces produced by the foil on the global axises of the domain as mentioned on previous calculation of this study. As it is seen on Figure 4.6 and

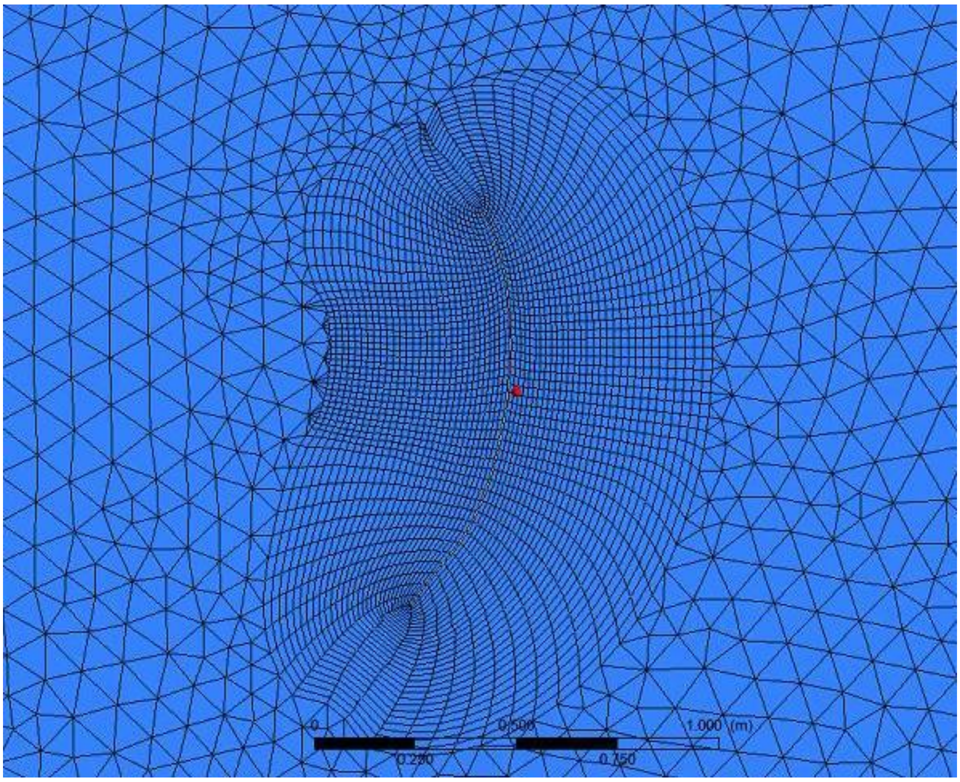

Figure 4.9: Inflation around the main sail.

Figure 4.7, X axis is in line with the centerline of the boat (from stern to the bow). Y axis is vertical to X axis as usual and the positive part of Y axis is on port side of the boat. So, the forces produced on X axis are showing the driving force. The produced forces on Y axis are showing the side force should not change because of practical experiences.

After several solution trials, the Y+ values become good. Expected values of Y+ are between 30 and 300. We have the values between app. 35 and 205. There are some very small areas which Y+ values are between 30 and 20.

Then, the calculations are made for different pumping angles and different frequencies for each angle. And the results are compared with the forces of non-pumping main sail which can be seen on Table A.2. The pressure contribution, velocity contours and Y+ can be seen on appendix.

Tangential force on X axis is the frictional part of drag force. The value of tangential force on Y is almost zero because the flow is coming to the sail almost vertical. Normal force on Y direction is showing the side force alone and it does not change with the frequency which is expected. The driving force is coming from the normal force on X axis. The results show that driving force is increasing with bot angle of pumping and frequency.

It is very clear that pumping the mainsail leads big performance gain to the boat. The calculations are made for 3 angles; 5, 10 and 15. Practically, you can not definitely know how many degrees you pump the main sail. Sailors say the target during the pumping is doing it as long as you can (they are trying to say big angles) and as fast as you can. First target is angle and second target is frequency. We have tried to keep the matrix on the limits of human being and seeing the differences between them. And we managed to get the same results with the practical sailor experiences.

The priority between the pumping parameteres can be explained as following. For example, the way of the mainsail sweep and angular velocity is same if you pump the main sail 1 time per second with 5 degrees or if you pump the main sail 1 time per 2 seconds with 10 degrees. However, you will get better performance with the high angle pumping.

The calculations are made for 2 type of motion; non pumping and continuously pumping. In reality, the sailors can make 3 pumps in a row with one wave. It is not allowed to pump without waves or more than 3 pumps in a row according to International sailing rules [21]. It is not managed to do the motion on that study same on real life, but the results can show the idea behind pumping the mainsail.

Theoretically, while pumping the main sail the incidence angle between the sail and wind increases and leads to get more force from the vortices. This is what is mentioned on the literature review about reverse von Karman street vortices on high angles of pitching. There is not a high angle of pitching exactly but it works like it because incidence angle on the start is also high.

As a result, it is obvious that pumping main sail brings more performance to the sailors. Although it is very hard and requires strength and condition, it causes to big differences between the competitors especially on one design races.

5. CONCLUSION

On this study, flow around a flapping foil is analyzed with CFX and compared to wind tunnel and numerical results. Then the CFX model is used to analyze the flow around the sail while it is pumped.

Most of the problems seen during the study were on the pre-process stage while preparing the model. Defining the motions on the program was not easy. There are several ways to do it and all of them are tried to get the similar movement of validation database. At the end, it is managed to make the foil move similar to validation data and calculations are started. It is called similar because the motion was not exactly same. The origin point of the motion was slighlty different.

On the second part of study, there was again problem about defining the motion of the sail. It is aimed to calculate the performance gain with three pumps which are between steady conditions of sail. However, the calculations are made by continuously pumped sails and non-pumped sails.

As a result, the origin of the pitching motion affect the results. As much as the origin goes away from the foil the thrust coefficient increases. It is obvious that frequency has a big effect on the thrust performance of flapping foil. Increasing the frequency make same affect on thrust and increases the thrust coefficient.

It has been seen that pumping the main sail affect the performance positively. Two parameters; angle and frequency are investigated on this study. The angle is more important than the frequency. The sailors have been advised always do as big pumps as they can do and then as fast as they can do.

5.1 Recommendation for future work

In order to calculate the exact performance gain through pumping the main sail the study has to be done for instant pumps. The study can be enlarge also by the number of pumps like 1,2 and 3. Moreover, the study can be done 3-D. So, 3-D effects of flow will be included. On the other hand, the model can be extended by adding also the boat and waves. The model will be more familiar to reality.

REFERENCES

[1] **Saydam, Z.** (2005). An Investigation into the Capabilities of Computational Fluid Dynamics in Sail performance Prediction, *Msc. Thesis,* University of Southampton, Southampton.

[2] **Marchaj, C. A.** (2000). Aero-Hydrodynamics of Sailing, Adlard Cloes Nautical, London

[3] **Claughton, A., Shenoi, A., Wellicome, J. (**2002). Sailing Yacht Design-Theory, Addison Wesley Longman Limited, Essex

[4] **Houghton, E. L. and Carpenter, P.W.** (2003). Aerodynamics for Engineering Students, Butterworth-Heinemann, Burlington

[5] **Depledge, B. H. (**2005). An Investigation into the use of High Roach Main Sails on International America's Cup Yacht Class, *MEngProject*, University of Southampton, Southampton

[6] **Larsson, L. and Eliasson, R. E.** (2000). Principles of Yacht Design, McGraw-Hill, Great Britain

[7] **Hearn, G. E. (**2004). High Performance Craft Lecture Notes, University of Southampton, Southampton

[8] **Lasher, W. C., Sonnenmeier, J. R., Forsman, D. R., Tomcho, J. (**2005). The Aerodynamics of Symmetric Spinnakers, The Pennsylvania State University, USA

[9] **Guglielmini, L. and Blondeaux, P. (**2008). Numerical Experiments on the Transient Motions of a Flapping Foil

[10] **Rozhdestvensky, K. V. and Ryzhov, V. A. (**2003). Aerohydrodynamics of Flapping-Wing Propulsors

[11] **Flores, M. D. (**2003). Flapping Motion of a Three-Dimensional Foil for Propulsion and Maneuvering of Underwater Vehicles

[12] **Alözkan, E. (**2006). Sürüklenme Açısı Altında Tekne Üzerine Gelen Kuvvetler Hesabı, *Yüksek Lisans Tezi,* İTÜ Fen Bilimleri Enstitüsü, İstanbul

[13] **Ramamurti, R. and Sandberg, W. (**2001). Simulation of Flow About Flapping Airfoils Using Finite Element Incompressible Flow Solver

[14] **Abbott, I. H. and Von Doenhoff, A. E.** (1959). Theory of Wing Sections, Courier Dover Publications

[15] (2006). ANSYS CFX-Solver Thory Guide

[16] (2004). Advanced Turbulence Notes, Fluent Europe LTD, Sheffield

[17] **Perez, S. E. (**2006). A First Course in Computational Fluid Dynamics

[18] User Manual of Star CCM+

[19] **WS Atkins Consultants.** Best Preactice Guidlines for Marine Applications of CFD

[20] **Ersahin, O. (**2010). CFD Analysis of the Flow Around the Sails

[21] **ISAF.** 2013, The Racing Rules of Sailing

APPENDICES

APPENDIX A: Post-process

APPENDIX A

Table A.1: Thrust coefficients of flapping foil.

K	Tangent X	mean Normal X	Total X	Ct
5	0.0248	0.004	0.0288	-0.010135991
6	0.0263	-0.004	0.0223	-0.007848354
8	0.0245	-0.0115	0.013	-0.004575274
11	0.0252	-0.031	-0.0058	0.002041276
15	0.0248	-0.075	-0.0502	0.017667596
20	0.0253	-0.1475	-0.1222	0.043007574

*Because of the coodinate system, while calculating thrust coefficient (-) has to be added in front of the total Force X.

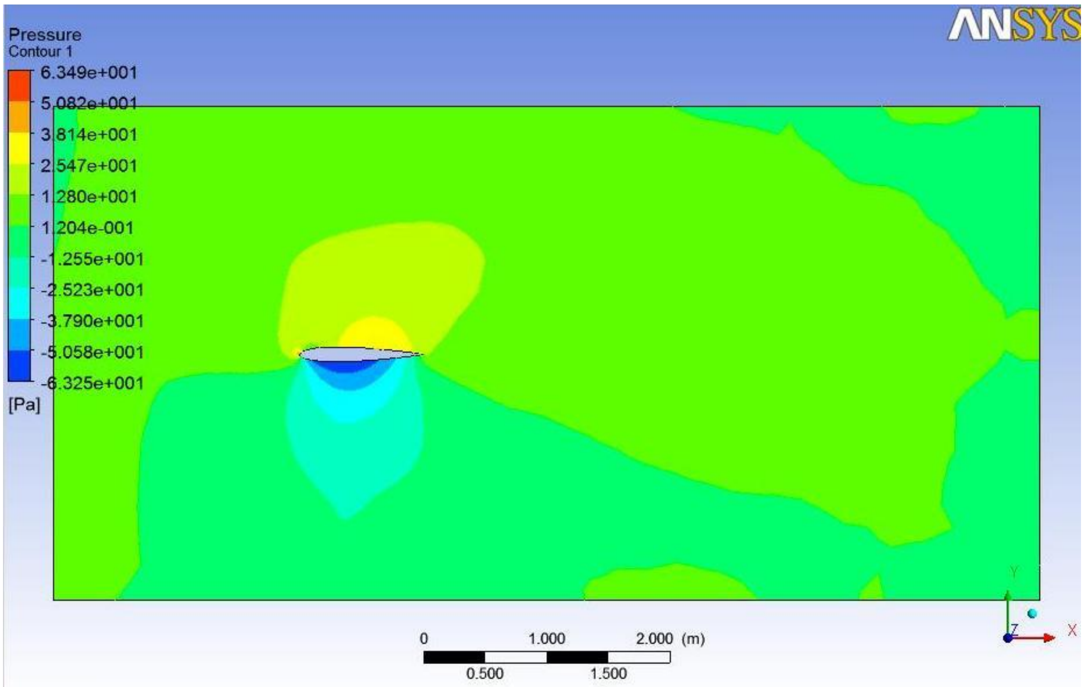

Figure A.1: Pressure contribution on flapping foil just after when the foil pitched 2^o.

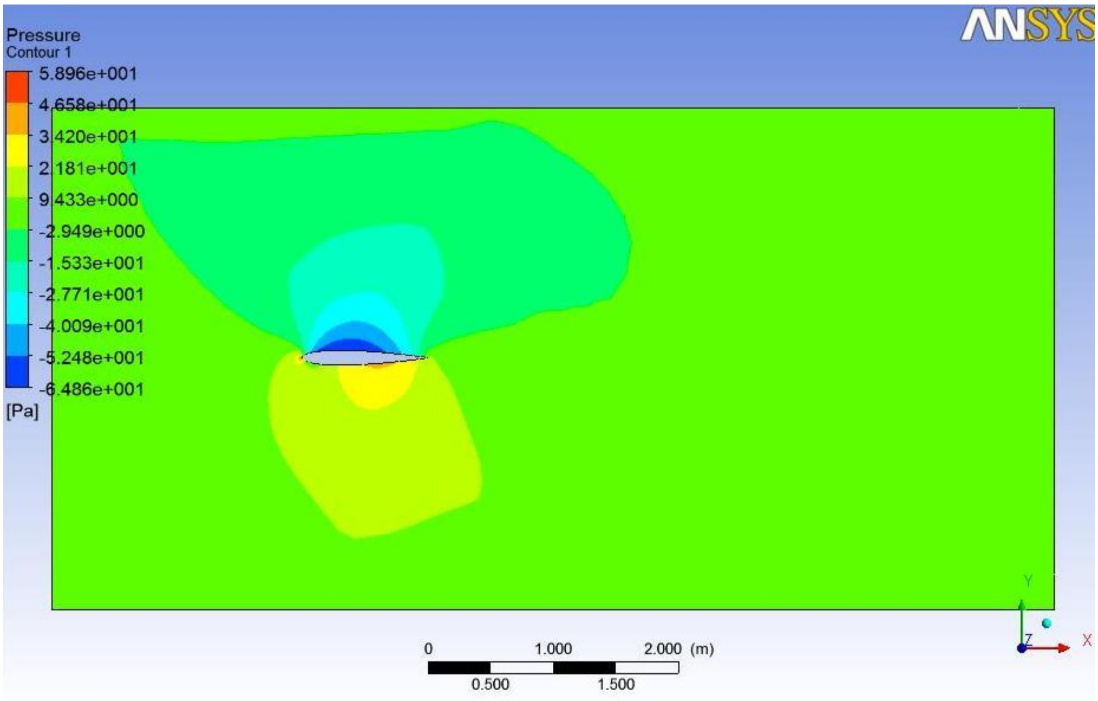

Figure A.2: Pressure contribution on flapping foil when the foil pitched -2^{o}.

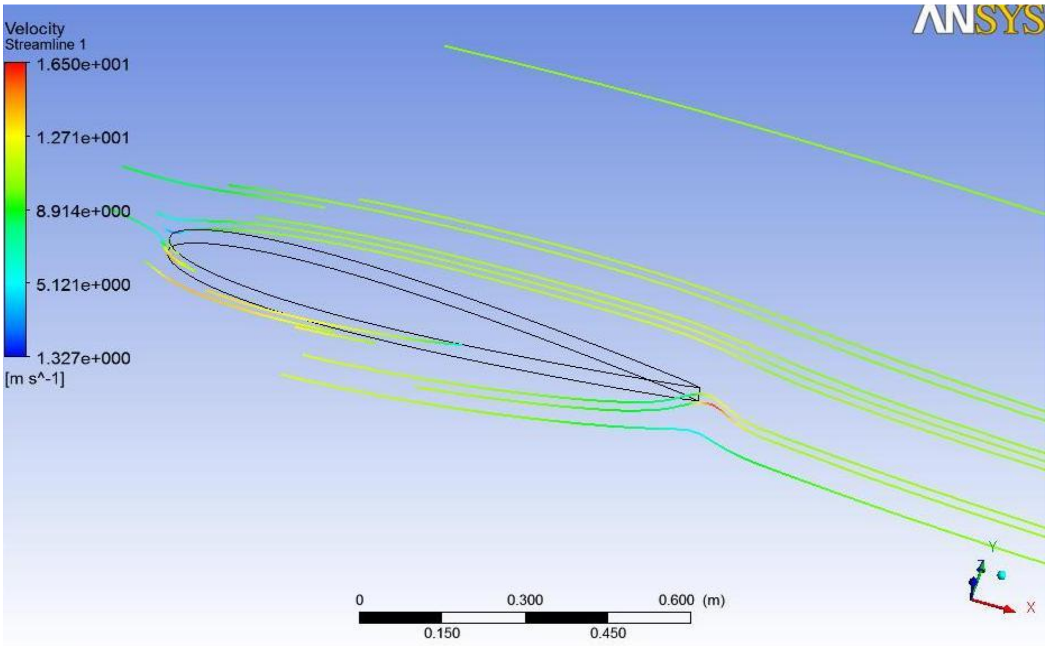

Figure A.3: Streamlines on flapping foil.

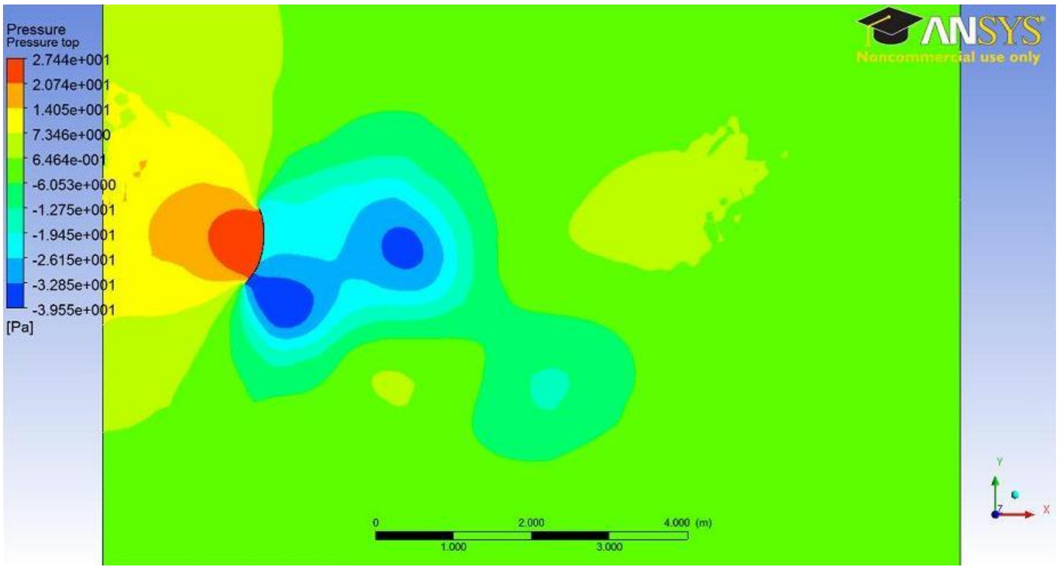

Figure A.4: Pressure contribution on main sail pumping.

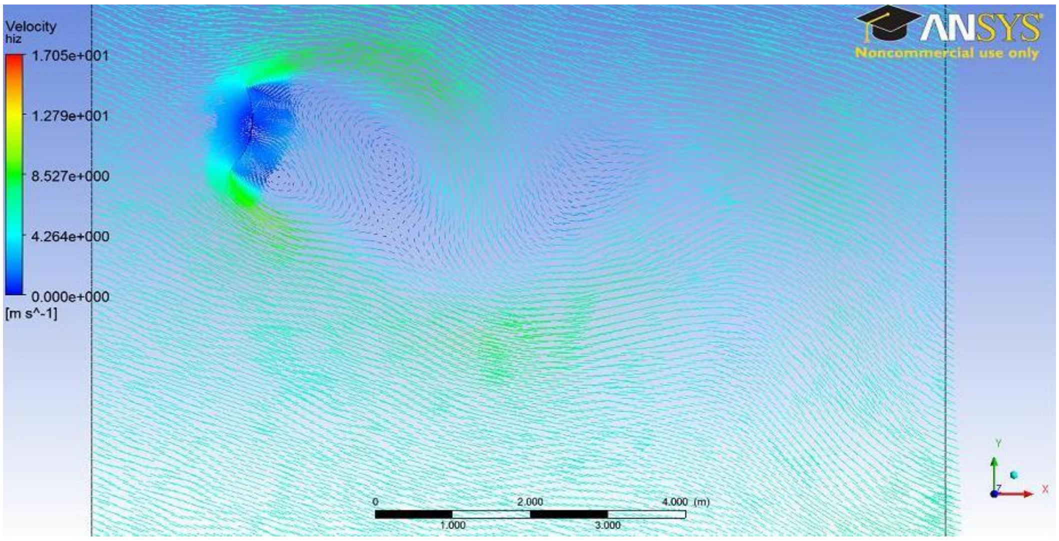

Figure A.5: Velocity contours on main sail pumping.

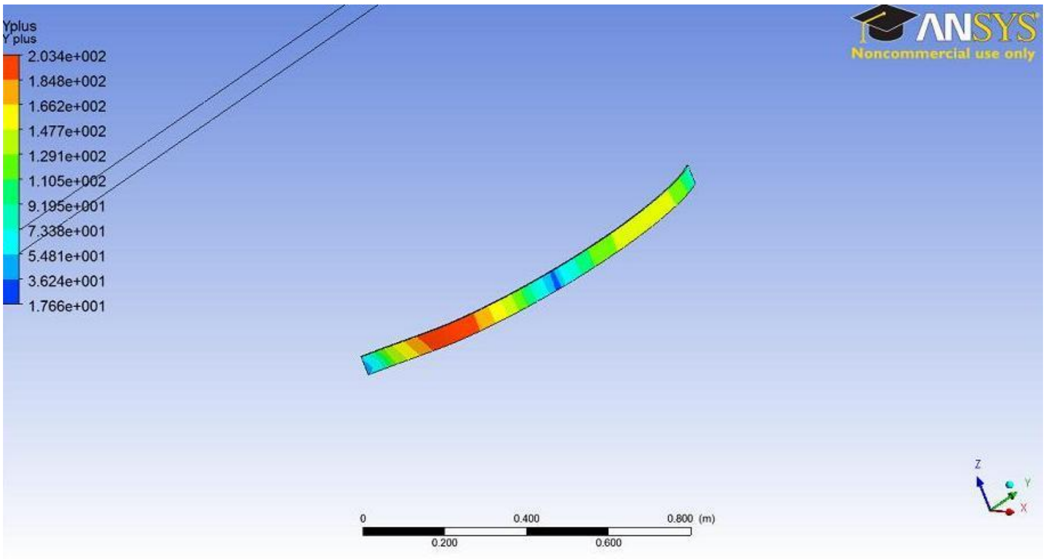

Figure A.6: Y+ values on main sail pumping.

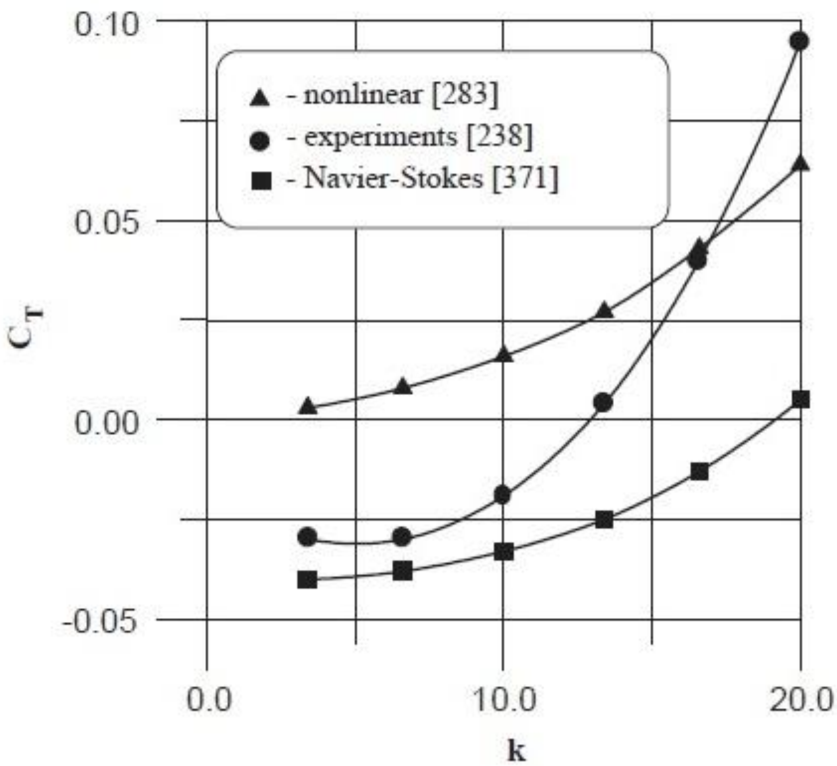

Figure A.7: Validation data [10].

Table A.2: Forces and performance gain on different pumping conditions.

Angle of Pumping	Frequency	Tangent X	Tangent Y	Normal X	Normal Y	Total X	Total Y	Performance Gain
0	-	-0.001	0	2.49	-0.44	2.489	-0.44	-
5	0.5	-0.001	0	2.6	-0.44	2.599	-0.44	4.42%
5	1	-0.001	0	2.63	-0.44	2.629	-0.44	5.62%
10	0.5	-0.001	0	2.68	-0.44	2.679	-0.44	7.63%
10	1	-0.001	0	2.74	-0.44	2.739	-0.44	10.04%
15	0.5	-0.001	0	2.8	-0.44	2.799	-0.44	12.45%
15	1	-0.001	0	2.85	-0.44	2.849	-0.44	14.46%

* Performance gain percentage is the rate of the difference between the Total X force of the trial and Total X force of non-pumping sail to the Total X force of non-pumping sail.

CURRICULUM VITAE

Name Surname: Osmancan ERŞAHİN

Place and Date of Birth: İzmir, 01.10.1986

Address: Suadiye Mah, Kadıköy - İstanbul

E-Mail: osmancanersahin@yahoo.com

B.Sc.: Naval Architecture, ITU

Professional Experience and Rewards:

-Best Dealer of Lagoon 2013
-Best Dealer of Lagoon 2012
-Working as sales manager of Lagoon catamarans and CNB custom yachts
(April 2011 -)
-Worked as second hand yachts brokerage responsible at Tezmarin
(January 2011- March 2011)
-Worked as project manager assistant on refit project of M/Y Zeepard (May 2010)
-Worked as a designer assistant on refit project of M/Y Talitha-G
(January 2008 – May 2008)

List of Publications and Patents:

Ersahin O., 2010, CFD Analysis of the Flow Around the Sails