

T.C
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

**NUMERICAL INVESTIGATION OF
FRONT WING ARRANGEMENTS
UNDER GROUND EFFECT**

METİN BAŞTUĞ

**A THESIS SUBMITTED FOR THE DEGREE OF
MASTER OF SCIENCE
DEPARTMENT OF MECHANICAL ENGINEERING**

GEBZE

2023

T.C.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

**NUMERICAL INVESTIGATION OF
FRONT WING ARRANGEMENTS
UNDER GROND EFFECT**

METİN BAŞTUĞ
**A THESIS SUBMITTED FOR THE DEGREE OF
MASTER OF SCIENCE**
DEPARTMENT OF MECHANICAL ENGINEERING

THESIS SUPERVISOR
ASSIST. PROF. DR. SEDAT TOKGÖZ

GEBZE
2023

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ

YER ETKİSİ ALTINDAKİ
ÖN KANAT DÜZENLEMELERİNİN
SAYISAL OLARAK İNCELENMESİ

METİN BAŞTUĞ
YÜKSEK LİSANS TEZİ
MAKİNE MÜHENDİSLİĞİ ANABİLİM DALI

DANIŞMANI
DR. ÖĞR. ÜYESİ SEDAT TOKGÖZ

GEBZE
2023



YÜKSEK LİSANS JÜRİ ONAY FORMU

GTÜ Fen Bilimleri Enstitüsü Yönetim Kurulu'nun 09/02/2023 tarih ve 2023/13 sayılı kararıyla oluşturulan jüri tarafından 07/03/2023 tarihinde tez savunma sınavı yapılan Metin Baştuğ'un tez çalışması Makine Mühendisliği Anabilim Dalında YÜKSEK LİSANS tezi olarak kabul edilmiştir.

JÜRİ

ÜYE
(TEZ DANIŞMANI) : Dr. Öğr. Üyesi Sedat Tokgöz

ÜYE : Dr. Öğr. Üyesi Mustafa Perçin

ÜYE : Dr. Öğr. Üyesi Recep Önler

ONAY

Gebze Teknik Üniversitesi Fen Bilimleri Enstitüsü Yönetim Kurulu'nun
...../...../..... tarih ve/..... sayılı kararı.

İMZA/MÜHÜR

SUMMARY

The role of aerodynamics has always been an important factor in motorsports. Since the front wing of a car faces the freestream first, the front wing plays the key role in controlling the flow on the downstream components and generating downforce. The front wing generates about thirty percent of the car's total downforce.

The scope of this study is to investigate the performance of a front wing using different arrangements. The effects of ground clearance, angle of attack and Reynolds number on the front wing is studied using numerical methods. In addition to that, the effectiveness of the vortex generators on the prevention of flow separation occurring with the increased angle of attack is observed.

Providing insight about the aerodynamics of a front wing under ground effect is the aim of this study. Generating higher downforce at higher ground clearances with increased angle of attack and vortex generators can be valuable for the wing design against the limiting regulations.

Key Words: Ground Effect, Vortex Generators, Front Wing, Aerodynamics.

ÖZET

Aerodinamiğin rolü, motor sporlarında her zaman önemli bir faktör olmuştur. Bir araçta serbest akış ile ilk karşılaşan kısım olmasından dolayı ön kanat, kanadın iz bölgesinde kalan kısımlardaki havayı kontrol eder ve aynı zamanda yere basma kuvveti oluşturur. Bir aracın toplam yere basma kuvvetinin yaklaşık olarak yüzde otuzunu ön kanat sağlar.

Bu çalışma ile kanadın farklı koşullar altındaki davranışının incelenmesi amaçlanmıştır. İncelenen parametreler, yerden yükseklik oranı, hücum açısı ve Reynolds sayısıdır. Bunlara ek olarak girdap üreticilerinin, yüksek atak açısına sahip kanatlarda oluşan akış ayrılmaları üzerindeki etkileri gözlemlenmiştir.

Bu çalışma ile yüksek hücum açısı ve girdap üreticileri kullanılarak, daha büyük yerden yükseklik oranlarında daha büyük yere basma kuvveti oluşturulabileceği görülmüştür. Bu durum, kanat tasarımları için değerli bir seçenek sağlayacaktır.

Anahtar Kelimeler: Yer etkisi, Girdap Üretici, Ön Kanat, Aerodinamik.

ACKNOWLEDGEMENTS

I would like to thank my thesis supervisor Asst. Professor Sedat Tokgöz for his consistent support and guidance during my studies. It would not be possible to finish this thesis without his patience and faith in me. I would also like to thank my family for believing in me, and providing mental and financial support. Lastly, I would like to express my gratitude to the Faculty of Engineering, Gebze Technical University, for providing me with resources and opportunity.



TABLE of CONTENTS

	<u>Page</u>
SUMMARY	v
ÖZET	vi
ACKNOWLEDGMENTS	vii
TABLE of CONTENTS	viii
LIST of ABBREVIATIONS and ACRONYMS	x
LIST of FIGURES	xi
LIST of TABLES	xiv
1. INTRODUCTION	1
1.1. Aerodynamic Background	1
1.2. Aerodynamics of Race Cars	4
1.3. Front Wing Aerodynamics	6
1.4. Aerodynamics of Vortex Generators	8
1.5. Overview and Objectives	9
2. LITERATURE REVIEW	10
2.1. Effect of Ground Clearance	10
2.2. Main Wing and Flap Angle of Attack	12
2.3. Reynolds Number	13
2.4. Wake Characteristics and Vortices	14
2.5. Boundary Layer Transition	16
3. METHODOLOGY	19
3.1. Geometry of the Wing	19
3.2. Mesh and Boundary Conditions	21
3.3. Governing Equations	23
3.4. Parameters	26
4. RESULTS AND DISCUSSION	28
4.1. Mesh Independence and Validation	28
4.2. Clean Wing at Low AoA (Phase 1 Cases)	30
4.3. Clean Wing at High AoA (Phase 2 Cases)	37

4.4. Separation Control with CtSVGs (Phase 3 Cases)	45
5. CONCLUSION AND FUTURE WORK	53
REFERENCES	54
BIOGRAPHY	59



LIST OF ABBREVIATIONS AND ACRONYMS

<u>Abbreviations</u>	<u>Explanations</u>
<u>and Acronyms</u>	
AoA	: Angle of attack
AR	: Aspect ratio
C_D	: Drag coefficient
C_L	: Lift coefficient
C_P	: Pressure coefficient
CFD	: Computational fluid dynamics
CoLVG	: Co-rotating large scale vortex generator
CoSVG	: Co-rotating sub-boundary layer vortex generator
CtLVG	: Counter-rotating large-scale vortex generator
CtSVG	: Counter-rotating sub-boundary layer vortex generator
DNS	: Direct numerical simulation
FIA	: Federation Internationale De'Automobile
LDA	: Laser Doppler anemometry
LES	: Large eddy simulation
LVG	: Large-scale vortex generator
PIV	: Particle image velocimetry
Re	: Reynolds number
SVG	: Sub-boundary layer vortex generator
T-S	: Tollmien-Schlichting waves
VG	: Vortex generator

LIST OF FIGURES

<u>Figure Nu:</u>	<u>Page</u>
1.1: Geometry of an airfoil.	2
1.2: Front wing of an F1 car.	7
1.3: Counter-rotating vortex generators on an aircraft wing.	8
2.1: Schematic of the separation-induced transition.	17
2.2: Schematic of the natural and bypass transition.	18
3.1: Definitions of the front wing.	20
3.2: Dimensions of the CtSVGs in millimeters.	21
3.3: Model of the front wing with vortex generators.	21
3.4: Domain and the refinement boxes.	22
3.5: Unstructured grid used for the simulations.	23
3.6: Unstructured grid used for the front wing.	23
4.1: Graph of the number of elements against the downforce coefficient.	29
4.2: Graph of the number of elements against the drag coefficient.	29
4.3: Pressure coefficient values at the symmetry plane.	30
4.4: Change of downforce with ground clearance.	32
4.5: Change of drag with ground clearance.	32
4.6: C_p of main wing at $2z/b = 0$ and $AoA = 14.6^\circ$.	33
4.7: C_p of flap at $2z/b = 0$ and $AoA = 14.6^\circ$.	33
4.8: Contours of velocity in x-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	34
4.9: Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$ and $2z/b = 0.92$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	34
4.10: Contours of vorticity in z-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$ (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	35
4.11: Contours of vorticity in x-direction viewed from downstream at $x/c = 1.053$ and $Re = 4.3 \times 10^5$ (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	36
4.12: Contour of vorticity in x-direction at $x/c = 0.526$, $x/c = 1.447$, $x/c = 2.368$, $x/c = 3.289$, $x/c = 4.21$ and $Re = 4.3 \times 10^5$.	36
4.13: Change of downforce with ground clearance.	38

4.14:	Change of drag with ground clearance.	38
4.15:	C_P of main wing at $2z/b = 0$ and $AoA = 22^\circ$.	39
4.16:	C_P of flap at $2z/b = 0$ and $AoA = 22^\circ$.	39
4.17:	Contours of velocity in x-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	40
4.18:	Contours of velocity in x-direction at $2z/b = 0$ and $h/c = 0.315$ (a) $Re = 4.3 \times 10^5$ (b) $Re = 2 \times 10^5$.	41
4.19:	Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$, $2z/b = 0.92$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	41
4.20:	Contours of vorticity in z-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$ (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	42
4.21:	Contours of vorticity in x-direction viewed from downstream at $x/c = 1.053$ and $Re = 4.3 \times 10^5$ (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.	43
4.22:	Contours of vorticity in x-direction at $x/c = 0.526$, $x/c = 1.447$, $x/c = 2.368$, $x/c = 3.289$, $x/c = 4.21$ and $Re = 4.3 \times 10^5$.	43
4.23:	Change of downforce with ground clearance at $Re = 4.3 \times 10^5$ for clean and CtSVG wing configurations.	45
4.24:	Change of drag with ground clearance at $Re = 4.3 \times 10^5$ for clean and CtSVG wing configurations.	46
4.25:	C_P of main wing at $2z/b = 0.044$, $Re = 4.3 \times 10^5$ and $AoA = 22^\circ$	47
4.26:	C_P of flap at $2z/b = 0.044$, $Re = 4.3 \times 10^5$ and $AoA = 22^\circ$.	47
4.27:	Contours of velocity in x-direction at $2z/b = 0.044$, $h/c = 0.211$ and $Re = 4.3 \times 10^5$ (a) Clean wing (b) Wing with CtSVGs.	48
4.28:	Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$, $2z/b = 0.92$ and $Re = 4.3 \times 10^5$ and $h/c = 0.211$ (a) Clean wing (b) Wing with CtSVGs.	49
4.29:	Contours of velocity in x-direction at $2z/b = 0.044$, $h/c = 0.315$ and $Re = 2 \times 10^5$ (a) Clean wing (b) Wing with CtSVGs.	50
4.30:	Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$, $2z/b = 0.92$ and $Re = 2 \times 10^5$ and $h/c = 0.315$ (a) Clean wing (b) Wing with CtSVGs.	50

- 4.31: Contours of vorticity in x-direction viewed from downstream at $h/c = 0.211$ and $Re = 4.3 \times 10^5$ (a) $x/c = 0.526$ (b) $x/c = 0.789$ (c) $x/c = 1.052$ (d) $x/c = 1.315$. 51



LIST OF TABLES

<u>Figure Nu:</u>	<u>Page</u>
3.1: The list of parameters used for the first, second and third phase cases.	27
4.1: Comparison of the C_D and C_L with the experimental results.	28



1. INTRODUCTION

Aerodynamics plays quite a significant role in the design of race cars, and this work aims to explain the physical phenomena occurring on a race car front wing. Even though the aerodynamics of race cars has been studied for decades, there is still much to investigate, especially due to frequently changing regulations.

There has been a strong emphasis on improving the performance of race cars by improving the engine, car shape, suspension systems, and tires since the early years of racing history. Low-drag race cars were used to achieve higher top speeds on a straight track during the initial years of motorsports. The importance of the downforce was recognized as the cars grew in speed, and as a result, more traction was required to be able to turn corners at a high speed. Increasing the downforce reduces the braking distance, and prevents wheel spin [Smith, 1978].

Opel RAK1 and RAK2 were the first cars to use inverted wings in an attempt to increase traction by generating downforce. However, a sharp increase in performance was observed in the late 1960s, which was the beginning of the aerodynamic era in racing history. The most substantial peak in performance is achieved by using aerodynamic devices such as front and rear wings in the 1970s [Katz, 1995].

The Federation Internationale De'Automobile (*FIA*) implemented the adoption of regulations over the years to ensure the safety of the driver as the cars became faster with the aerodynamic developments. Currently, a modern race car can generate a lateral acceleration of 5G under braking, and 3G under acceleration [Roberts, 2013].

1.1. Aerodynamic Background

In order to have a better understanding of race car aerodynamics, the basics of aerodynamics will be discussed in this section. Airfoils are used in almost all aerodynamic devices, and therefore airfoil aerodynamics play a crucial role in a race car as well. The geometry of an airfoil can be found in Figure 1.1 [Kundu

and Cohen, 2008]. The terminology used for defining an airfoil can be summarized as [Gunston, 2005]:

- Leading edge: Front edge of an airfoil that faces the free stream
- Trailing edge: Rear edge of an airfoil
- Chord: Straight line joining the leading edge and the trailing edge of an airfoil
- Camber: The line that is located between the centerline of the upper and lower surfaces
- Thickness: Maximum distance between the upper and lower surfaces of an airfoil perpendicular to the chord line
- Angle of attack (α): Angle between the chord and flow direction
- Pressure side: Side of an airfoil that has higher pressure. The lower surface is the pressure side for an aircraft wing, whereas the upper surface is the pressure side for a race car wing
- Suction side: Side of an airfoil that has lower pressure. The upper surface is the suction side for an aircraft wing, whereas the lower surface is the suction side for a race car wing

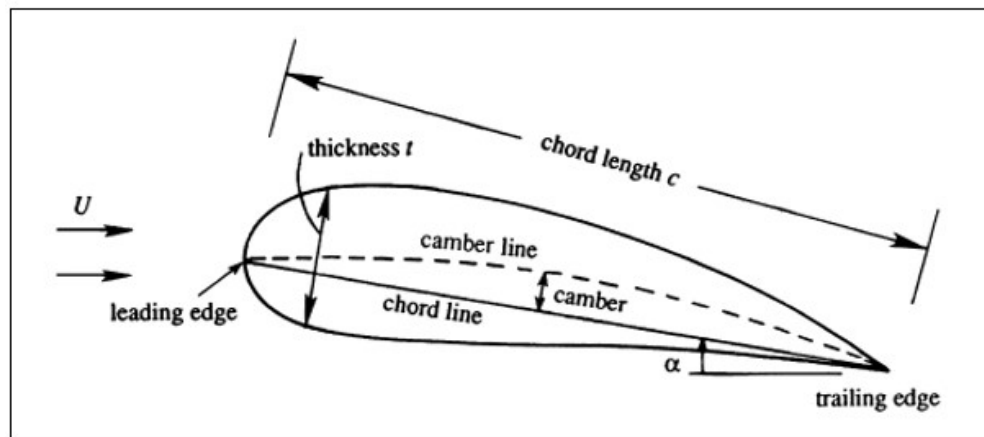


Figure 1.1: Geometry of an airfoil.

The special shape of the airfoil creates pressure difference between the pressure and suction sides which generates lift (or downforce depending on the orientation) using Bernoulli's principle. Lift is the force acting on the wing in the opposite direction of gravity, whereas downforce acts in the same direction as

gravity. The downforce coefficient is used in this study due to the use of inverted wings. The lift coefficient (C_L) is defined as [Anderson, 2007]:

$$C_L = \frac{L}{\rho_{\infty} V_{\infty}^2 S/2} \quad (1.1)$$

where ρ_{∞} is the freestream density, V_{∞} is the freestream velocity, S is the planform area and L is the lift force.

Drag is the force that is directed in the opposite direction of the motion of the object, and it resists the motion of the object [Bertin and Cummings, 2009]. The drag coefficient (C_D) is defined as:

$$C_D = \frac{D}{\rho_{\infty} V_{\infty}^2 S/2} \quad (1.2)$$

In the case of subsonic flows, total drag is a combination of skin-friction drag, pressure drag and lift-induced drag. The viscous effects of the fluid cause skin friction drag. Skin-friction drag is lower in laminar flow, and airfoils that promote laminar flow can be used to decrease skin-friction drag. In general, the shape of the body is the most important factor in the pressure drag. Streamlining the body prevents flow separation, and causes less pressure drag. Induced drag is an aerodynamic force that occurs when the direction of the flow is redirected by an object. The finite wings of an aircraft are one of the most known examples of induced drag. On an aircraft wing, the air escapes from the pressure side to the suction side at the wingtips due to the pressure difference. This motion of the fluid disturbs the chord wise flow. This change in direction is a result of the induced drag [McLean, 2012].

Flow separation is a phenomenon that reduces the lift and increases the pressure drag of an aerodynamic device drastically. The boundary layer separates from the surface of an object forming a wake, referred to as flow separation [White, 2011]. The boundary layer is a thin layer close to the wall in which velocity transition from zero to a finite value occurs. The no-slip condition causes the velocity to be zero at the wall. As the distance from the wall increases, the velocity increases as well until it reaches the freestream value [Schlichting and

Gersten, 2000]. Flowing against an increasing pressure decreases the momentum of the flow, and eventually reverses the direction of the boundary layer flow as well. As a result, the flow is detached from the surface and forms vortices. In a boundary layer flow, the momentum transfer in the wall normal direction can prevent the flow separation, and this momentum transfer depends on the Reynolds number.

Flows can be categorized as laminar or turbulent depending on their characteristics. Reynolds number (Re) is a dimensionless number that defines the turbulence characteristics of a flow. It is defined as the ratio of the inertial forces to the viscous forces:

$$Re = \frac{\rho_{\infty} V_{\infty} L}{\mu_{\infty}} \quad (1.3)$$

where μ_{∞} is the freestream viscosity, and L is the characteristic length which is the length of the chord line [Flandro et al., 2011]. Reynolds number influences the boundary layer, and boundary layer thickness decreases as the Reynolds number is increased.

1.2. Aerodynamics of Race Cars

A race car consists of different components which exhibit certain aerodynamic behaviors. Those components can be listed as:

- Car body
- Diffuser
- Wheels
- Rear wing
- Front wing
- Vortex generators

The engine was the main research area in order to improve the performance of a race car. After the initial performance boost by the engine improvements, manufacturers recognized the importance of aerodynamics and car body shape.

One of the first examples of a streamlined race car was the 1899 Camille Jenatzy, which had a specific body shape to reduce the aerodynamic drag. Another car design was the 1924 Tropfenwagen, which had a shape similar to an airfoil shape to reduce drag as well. The importance of downforce was not recognized until the race car Chaparral 2C was developed in 1965 [Katz, 2006].

Diffusers are used to increase downforce, and they consist of underbody channels that increase the speed of the flow entering the device by constraining the flow area. This is followed by a drop in speed by the expanding flow region without causing flow separation. George [1981] studied the effect of ground clearance on downforce using a diffuser angle of 10° , and found that as the ground clearance is decreased, downforce increases until it hits the critical peak value.

There are two types of race cars in terms of wheel structure which are open-wheel and closed-wheel cars. Open wheels are affected by the upstream components, and they also affect downstream components directly. Therefore, open-wheel aerodynamics is more influential on vehicle performance. The wheel is a source of high drag on a race car, and therefore designers redirect the air in order to avoid the flowing on a wheel. The rotation of the wheel should also be considered to evaluate the aerodynamic properties of the wheel, and it was observed that rotating wheels produce lower lift and drag coefficients than stationary wheels [Fackrell and Harvey, 1974].

Rear wings are used to increase the traction of the rear wheels, and therefore increase the cornering speeds. They can also be used in combination with a diffuser to improve the downforce even further depending on the diffuser angle. Hu et al. [2011] showed that reduced diffuser angle results in an increase in the lift coefficient. Aerodynamics of front wings and vortex generators are explained in the following sections.

Downforce increases the cornering speed and the tire normal force, which prevents sliding. Tire adhesion combined with downforce can change the turning characteristics of a race car. The turning angle of the front axle is defined as the Ackermann angle. When the turning angle of the vehicle is less than the Ackermann angle, the condition of understeer occurs. By increasing the downforce on the back side of the vehicle, understeer can be avoided. When the vehicle turns more than the Ackermann angle, oversteer occurs, and downforce

on the front side can be increased in return [Mitchell et al., 2006]. When defining a vehicle's turning characteristics, the front/rear downforce ratio is a deciding factor. The front/rear downforce ratio and the total downforce of the car define the stiffness of the suspensions. The front/rear downforce ratio can change due to disturbances, leading to undesirable oversteering during a turn that can spin the car. By increasing the suspension stiffness, the pitch sensitivity of the vehicle can be reduced, and this problem can be corrected.

The shape of the track affects the configuration of the car. High downforce or low drag setup can be used depending on the number of high-speed and low-speed corners. A track with a high number of high-speed corners requires a low drag setup to increase maximum speed. A rearward center of pressure can be used to have an understeer characteristic and provide stability during turns on a track with many high-speed corners. A high downforce setup can be used for low-speed corners to turn faster and a forward center of pressure gives an oversteer characteristic that increases the turning rate of the car as well [Katz, 1995].

1.3. Front Wing Aerodynamics

The front wing of a race car, which can be found in Figure 1.2 [Partridge, 2021], consists of airfoils which are also used on aircraft; however, there are differences between aircraft and race car wings. Inverted airfoils are used on race car wings to generate negative lift, whereas aircraft wings generate positive lift. Due to regulations, race car wings have a smaller aspect ratio(AR), the ratio of the wing span to the length of the chord. The ground effect which is explained in detail in Title 0 plays a vital role in race car wings, and aircraft wings do not operate in the ground effect except landing or taking off. Another difference is that race cars usually operate in the leading car's wake, which changes the drag and decreases the downforce coefficients [Soso and Wilson, 2006].



Figure 1.2: Front wing of an F1 car.

25-30% of a race car's downforce is generated by the front wing [Agathangelou and Gascoyne, 1998]. However, maximizing the downforce of a front wing is not the main priority when designing a race car. The front wing controls the flow on the downstream components and should direct the flow in the desired way to increase performance. The central portion of the front wing should maximize the flow going to the underbody and the diffuser. Bluff objects such as tires are a source of positive lift and high drag. The front wing provides outwash to move the air away from the wheels and decrease energy losses. Similarly, the front wing directs the turbulent air into the radiators to help with cooling.

The components of a race car are close to each other, and therefore the interaction between those components changes the aerodynamic behavior significantly. However, analyzing all of the components of a race car together has a demanding computational cost, and therefore only the front wing of a race car is investigated in this study. This approach is used in the literature as well [Zerihan, 2001].

The front wing is the first component that meets the air. It is quite challenging to test the front wing due to the difference in conditions between a wind tunnel and a track. There are higher turbulence conditions on track due to driving in the leading car's wake, whereas the flow has high energy and low turbulence intensity in a wind tunnel. The aerodynamic analysis of an open-wheel car front wing in wind tunnel conditions is carried out in this study.

1.4. Aerodynamics of Vortex Generators

Vortex generators are aerodynamic devices which are effective in eliminating or reducing flow separation. They consist of small vanes which are attached close to the leading edge of the suction side of the wing to control the flow. Around those vanes, streamwise vortices are created, which transfer momentum between the freestream and the boundary layer. [Kuya et al., 2009]. Vortex generators can have different configurations, sizes and shapes. Counter-rotating sub-boundary layer vortex generator (*CtSVG*), counter-rotating large-scale vortex generator (*CtLVG*), co-rotating sub-boundary layer vortex generator (*CoSVG*) and co-rotating large-scale vortex generator (*CoLVG*) are the types of vortex generators which are used in motorsports commonly.

Device height is a critical factor for a vortex generator. Sub-boundary layer vortex generators (*SVG*) have a height of 10% - 50% of the boundary layer thickness, and large-scale vortex generators (*LVG*) have a height higher than 50% of the boundary layer thickness. Optimally placed SVGs can be as effective as LVGs in terms of separation control and generate less drag [Lin, 2002]. The strength of the vortices generated by VGs can be adjusted by changing the angle between the device and the air flow. The counter-rotating vortex generators on an aircraft wing can be found in Figure 1.3 [Tallman, 2017].



Figure 1.3: Counter-rotating vortex generators on an aircraft wing.

1.5. Overview and Objectives

The purpose of this study is to monitor the behavior of employing various factors, including ground clearance, Reynolds number, and angle of attack (AoA). The analyses which yield flow separation at the trailing edge of the wing are used for the implementation of the vortex generators.

The study consists of three main sections. The first section shows the flow behavior when there is no flow separation, and conditions are close to ideal. An AoA of 14.6° is used in the first section, which is in line with the validation case. The Reynolds number and ground clearance effects are observed.

The second section includes cases with flow separation. The AoA is increased to 22° to observe flow separation. The dependencies of Reynolds number and ground clearance change with the flow separation. Cases with and without flow separation are compared at the end of the second section in order to investigate the effects of AoA .

In the last section, the cases with an AoA of 22° are used, and counter-rotating large-scale vortex generators are implemented in order to prevent the flow separation. The aerodynamic improvements in the downforce generation with the CtSVGs are observed.

2. LITERATURE REVIEW

The front wing of a race car has been an essential research topic for decades due to its role in providing downforce and controlling the flow direction downstream. Factors that affect the aerodynamics of a front wing can be listed as:

- Ground clearance
- Angle of attack and flap deflection
- Reynolds number
- Wake characteristics and vortices
- Boundary layer transition

A detailed literature review of each factor will be discussed in the following sections.

2.1. Effect of Ground Clearance

Single-element front wings at small ground clearances were used on race cars initially. However, it was discovered in the early 1980s that lowering the ground clearance increases downforce because of the restricted flow under the inverted wing. Since the flow is constrained under the wing, the flow moves faster to satisfy continuity, and therefore the air has lower pressure due to Bernoulli's principle [Dominy, 1992].

Literature shows the influence of ground effect on single and multi-element front wings using experiment and computational methods [Ranzenbach et al., 1997; Ranzenbach and Barlow, 1994, 1995, 1996]. Two-dimensional studies were carried out on symmetric NACA 0015 and negatively cambered NACA4412 airfoils. In those studies, the Reynolds number and angle of attack were held constant to focus on the effect of the ground clearance. It was shown that as the ground clearance decreases, downforce rises until it reaches its maximum value and if the ground clearance is decreased further, downforce starts declining. This phenomenon is called force reversal, and it occurs at a ground

clearance smaller than the 10% of the chord length. The force reversal is attributed to the merging of the wing and ground boundary layers. Furthermore, it is stated that force reversal occurs at a higher ground clearance when the camber of the airfoil is increased.

Similarly, Zerihan and Zhang [2000] conducted experimental and numerical studies about the effects of the ground clearance on a single-element front wing. It was confirmed that more downforce is generated as the ground clearance is decreased up to the critical ground clearance. Those regions are called force-enhancement and force-reduction regions, respectively. It was shown that a trailing edge separation develops in the force-reduction region. As the ground clearance is decreased further, this separation moves towards the leading edge.

Zhang and Zerihan's [2003, 2004] research of a multi-element configuration revealed that the majority of the downforce is produced by the main wing. Stronger tip vortices were observed compared to the single-element configuration, and those vortices significantly affect the downforce generation. Due to flow restriction and concentrated vortices in the force-enhancement region, the downforce increases as the ground clearance is reduced. If the ground clearance is decreased further, the breakdown of the tip vortices occurs, but downforce still increases at a lower rate until a critical ground clearance is reached. Below the critical ground clearance, trailing edge separation occurs, which causes a reduction in downforce and an increase in drag.

Mokhtar [2005] conducted a study about the ground effect, and four different airfoils were tested. Initially, the ideal ranges of the angle of attack, ground clearance, and Reynolds number are found using XFOIL. After ideal ranges were found, CFL3D was used to examine how the ground clearance affects the aerodynamic properties. It was shown that drag and downforce increase as the ground clearance is decreased, and ground clearance is effective at heights less than $h/c = 0.6$, where h is the distance from the ground and c is the chord length.

In summary, ground clearance has a significant effect on downforce. Downforce increases as the ground clearance decreases until a certain point below which trailing edge separation occurs. An increase in drag and a decrease

in downforce occurs in the force reduction region, which is at heights below the critical ground clearance.

2.2. Main Wing and Flap Angle of Attack

As mentioned, the angle between the airfoil's chord line and the direction of the freestream flow is known as the angle of attack. Up to a certain limit where flow separation occurs, the lift coefficient increases with the angle of attack. Leading edge separation occurs on thin airfoils when the camber or the angle of attack is high, whereas trailing edge separation occurs on thicker airfoils. [Katz, 1995].

Zerihan and Zhang studied the effect of incidence on a single-element wing. A maximum downforce coefficient of -1.35 was seen at an incidence of 11.3 degrees. The results showed that when the incidence is increased above 11.3 degrees, flow separation occurring at the trailing edge moves upstream. When the incidence is increased even further, the flow separation was seen at the leading edge of the wing [Zerihan and Zhang, 2000].

To examine how the angle of attack affects the performance of the wing, numerical approaches are also used. A 2-D numerical study was conducted on the UIUC700 multi-element profile by Soso and Wilson [2004]. It was noted that as the angle of attack of the main wing increases, so does the ground clearance at which the maximum lift coefficient is reached. The thickness of the wake also increases with the angle of attack of the main wing.

A flap on an aircraft is a high-lift device which is attached at the trailing edge of the wing, whereas flap on a race car is a small wing placed behind the main wing to increase the downforce. The only way to alter the performance of a multi-element front wing during a race without changing the wing entirely is to adjust the flap AoA. Experimental measurements on a UIUC700 multi-element wing configuration showed that increasing the flap AoA by 10 degrees increases the lift coefficient by 0.5. There was no significant change in drag by changing the flap AoA. The main wing AoA was kept constant throughout those measurements [Jasinski and Selig, 1998]. Soso and Wilson [2004] also studied the effect of flap AoA on a multi-element front wing. Similarly, it was shown that

the lift coefficient increases, and small changes in drag are seen with the flap AoA. In addition, the downforce and drag curves are only displaced upwards by increasing the flap AoA. This means C_L -ground clearance or C_D -ground clearance curves show a similar trend to flap AoA changes.

In conclusion, when the angle of attack increases in the force-enhancement region, downforce generation increases. However, the critical ground clearance increases as the angle of attack increases.

2.3. Reynolds Number

The effect of the Reynolds number and other significant parameters on a multi-element front wing was investigated by Jasinski and Selig [1998]. When the Reynolds number is increased from 0.7×10^6 to 1.1×10^6 , the average negative lift coefficient increases by 2.5%, and the average drag coefficient decreases by 2.3%.

A study on the Reynolds number sensitivity of single and multiple-element wings was done by Roberts et al. [2016]. For the single-element wing, there is no significant change in downforce as the ground clearance is decreased at a Reynolds number of 1.63×10^5 . Downforce remains constant until the force reduction region is reached. Force-enhancement and force-reduction regions are evident at a Reynolds number of 2.24×10^5 . As the Reynolds number is increased, the dependence on the downforce and drag at a given ground clearance decreases. For the multi-element wing, there is a slight variation of downforce with Reynolds number in the force-enhancement region with concentrated vortices. As the ground clearance is decreased further, the Reynolds number significantly affects the downforce generation. Another observation was that the critical ground clearance is lowered as the Reynolds number increases. In terms of drag coefficient, it was seen that the Reynolds number dependency is relatively small.

In conclusion, Reynolds number sensitivity is higher in the force-reduction region. Relatively small changes in drag and downforce occur in the force-enhancement region as the Reynolds number changes.

2.4. Wake Characteristics and Vortices

Wake characteristics and vortices play a vital role in the performance of a race car. The changes in the technical regulations about race cars can increase the chances of overtaking drastically by reducing the negative effects of the wake, namely the decreasing in downforce and increase in drag. The front wing of a race car dictates the flow of the downstream components; therefore, the wake characteristics of a front wing is an essential research topic. A study by Zerihan and Zhang [2002] shows that at large ground clearances, the front wing produces a small and less turbulent wake. As this wake travels downstream, it moves upwards. Large ground clearances prove to be beneficial for the performance of the downstream components; however, there is less downforce generation at large ground clearances. The size and turbulence level of the wake increase as the ground clearance is reduced. Another observation about the reduction of the ground clearance is the downward motion of the wake as it develops. Particle image velocimetry (*PIV*) measurements reveal the flapping motion of the wake moving downstream. A high lift coefficient can be obtained with small ground clearance with the disadvantage of having a highly turbulent and large wake.

In a race, it is common to see a race car moving in the wake of the leading car, leading to aerodynamic losses on the front wing. Soso and Wilson [2006] studied this phenomenon using a bluff body and a single-element front wing in a wind tunnel. It was shown that there is a decrease in downforce and an increase in drag on the downstream body, and the loss in downforce increases with higher ride heights. The loss of downforce was higher at the mid-span than at the tips. Additionally, the impact of the separation between the front wing and the bluff body was investigated. As the distance between those two bodies was decreased, the downforce generated by the front wing decreased. Another interesting observation was that the downforce produced by the front wing behind a bluff body could be higher than the freestream case when the distance between two bodies is at a certain point.

Wilson et al. [2008] conducted a similar study to investigate the effects of drafting on the wake region. The previous study of Soso and Wilson on the same subject had wind tunnel size limitations, and a short bluff body was used to

address this issue. High turbulence, velocity deficit and change of incidence due to the trailing vortices are the essential characteristics of the wake behind a bluff body. The wind tunnel measurements showed that the intensity of the wake decreases with the lateral distance between bodies. However, the strength of the vortices persists for longer distances. The downstream body also experiences a delay of stall onset by 5 degrees which is attributed to the highly turbulent wake or change of wing incidence due to vortices.

Garrod [2004] analyzed the tip-vortices of a simplified front wing and observed the difference between the moving and fixed ground. A concentrated vortex core was observed when a fixed ground was used. Due to the vortex breakdown, no vortex core was identified for the moving ground case. The conclusion is that the vortex breakdown occurs at a higher ground clearance when moving ground is enabled. Pegrum [2006] studied the same phenomenon using a complex McLaren front wing, and the difference in vortex strength was observed between the moving and fixed ground cases. However, vortex core formations were similar in those two cases, which contradicts the test results of the single-element wing case of Garrod. It can be deduced that the geometry of the front wing manipulates the vortex core formation. It was also shown that the strength of the vortices was affected by the ground clearance.

The effect of the endplate size and ground clearance on the tip vortices is studied by Galoul and Barber [2007]. Laser Doppler Anemometry (*LDA*) measurements showed the co-rotating vortices on the bottom and top of the endplate. The upper vortex changes its direction because of the lower vortex, which was determined to be stronger.

Flow physics of a front wing with vortex generators is studied numerically and experimentally by Kuya et al. [2009, 2010]. In the middle of CtVGs, the flow separation is suppressed, according to the measurements of flow visualizations and off-surface flow. Horseshoe vortices were formed when the CtLVGs were used, and the same phenomenon occurred when the separation between the CtSVGs was increased. The results suggested that CoVGs were not effective in preventing flow separation. Wake surveys show that a slight variation in the spanwise direction was observed when the CtSVGs were used, whereas the variance in the spanwise direction increased with the use of CtLVGs. Computational fluid dynamics (*CFD*) simulations using Spalart-Allmaras

turbulence model were also used to validate the experimental test results. Flow survey results showed that the vortices generated by the CtSVGs break down, while the vortices generated by the CtLVGs persist for a longer distance downstream. The vortices generated by the CoVGs dissipate quicker; therefore, CoVGs proved to be ineffective in preventing flow separation. Experimental and numerical test results suggested that CtSVG is the most effective VG setup to prevent flow separation.

The wake characteristics and vortices play a vital role in the performance of a race car. The changes in technical regulations about race cars can increase the chances of overtaking drastically by reducing the negative effects of the wake, namely the decrease in downforce and increase in drag.

2.5. Boundary Layer Transition

Due to its importance in preventing flow separation, the laminar-turbulent transition is one of the deciding factors which affects the flow on the airfoil. There are three types of laminar-turbulent transitions: natural transition, separation-induced transition and bypass transition.

Natural transition is based on the stability theory, which states that if the disturbances in the flow do not decay, the flow becomes turbulent. The disturbances decay or grow stronger depending on their wavelength and frequency, and if the disturbances grow as they move downstream, Tollmien-Schlichting ($T - S$) waves are formed. Experimental research about the oscillations of a laminar boundary layer on a flat plate was done by Schubauer and Skramstad [1947], and the conclusion was that Tollmien-Schlichting waves with an external or internal source cause transition on a flat plate. As the laminar-turbulent transition occurs, Tollmien-Schlichting waves form vortices. Those vortices break down, and the turbulent spots created by the vortex breakdown merge to form a completely turbulent flow.

Turbulence increases the momentum diffusion in the normal direction to the flow and makes the boundary layer stronger against the adverse pressure gradients. The laminar boundary layer is unable to overcome the adverse pressure gradients at lower Reynolds numbers. Separation-induced transition

consequently occurs, and a separation bubble is formed. The schematic of the separation-induced transition can be found in Figure 2.1 [Roberts, 2013]. The separation bubble's size or existence depends on the Reynolds number. A flow with a high Reynolds number can overcome the adverse pressure gradients, so the separation bubble becomes smaller or disappears [Tani, 1964]. The separation bubble becomes larger as the adverse pressure gradients increase, and full separation may occur if the adverse pressure gradients are increased further. The separation bubble is associated with a region of constant pressure. After the separation bubble is formed, Kelvin-Helmholtz instabilities initiate turbulence generation, and flow reattachment occurs due to the vortices forcing the flow in the normal direction to the wall. This phenomenon was observed with Large Eddy Simulations [Yang and Voke, 2001] and PIV measurements [Hain et al., 2009].

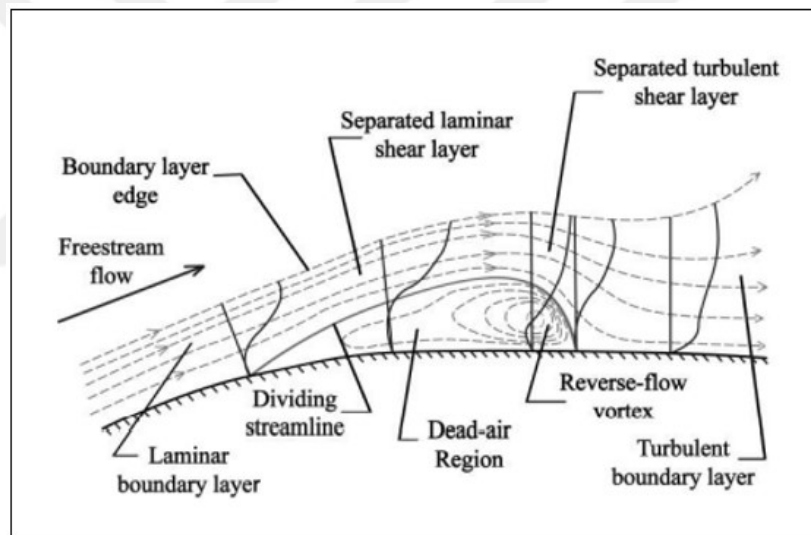


Figure 2.1: Schematic of the separation-induced transition.

Another type of laminar-turbulent transition is bypass transition which can be achieved by using vibrations or boundary layer trips. Turbulent spots appear directly in the flow, and the natural transition's initial stages are bypassed. Therefore, stability theory does not apply to the bypass transition mechanism. Zaki [2013] investigated the bypass transition using Direct Numerical Simulations (*DNS*) and linear theory. It was shown that low-frequency perturbations could penetrate the boundary layer. Another observation was the presence of the inner and outer instabilities, which occur at the boundary layer

edge and the wall, respectively. The schematic of the natural and bypass transition can be found in Figure 2.2 [Schlichting and Gersten, 2000].

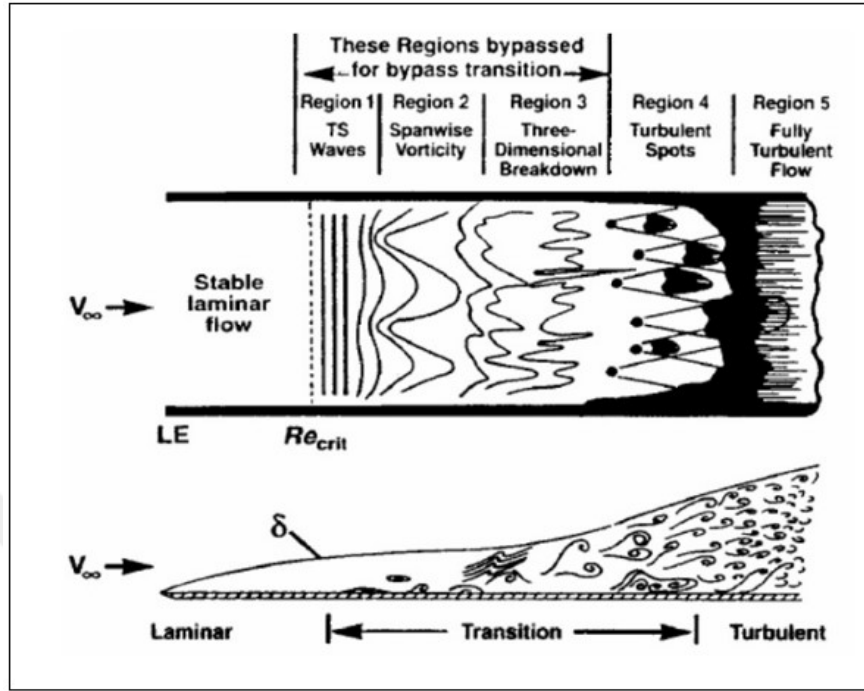


Figure 2.2: Schematic of the natural and bypass transition.

Turbulence models such as $k-\omega$ and $k-\varepsilon$ are used for fully turbulent flows, and struggle with finding the transition onset. Large Eddy Simulations (*LES*) and Direct Numerical Simulations can predict transitional phenomena correctly; however, the required computational cost and time make those methods impractical. Transition-turbulence models require moderate computational resources and can predict the transition effects. The $\gamma-Re_\theta$ turbulence model [Menter et al., 2006] uses empirical correlations to determine the transition, but it is difficult to find those correlations for different cases. $k-k_L-\omega$ turbulence model by Walters and Cokljat [2008] can solve the low Reynolds number cases accurately, and empirical correlations are not required to predict the transition effects.

Considering the transition effects on a front wing is vital because it alters the pressure distribution, directly determining the performance. Proper use of the turbulence models can reduce the computational cost and predict the transitional phenomena accurately.

3. METHODOLOGY

The literature review on the front wings shows that there is constant aerodynamic development in the world of race cars. After the literature was reviewed, it was observed that there is a lack of studies about the separation control devices on a multi-element wing under ground effect. This study aims to build a structure to explain the behavior of the front wing under certain conditions. After the investigation of the front wing under certain conditions is finished, flow separation devices such as VGs are used to prevent flow separation.

The present study focuses on analyzing a race car front wing using appropriate computational models. The analyses are performed using the CFD software FLUENT. To represent realistic results, the effects of the ground on the front wing are also considered. A total of 18 cases are presented in three phases.

3.1. Geometry of the Wing

The front wing used in this study is a double-element front wing consisting of a main wing and flap. The airfoil profile for the wing is NASA GA(W), type LS(1)-0413 MOD. The wing has a chord length of 223.4 mm for the main wing and 165.7 mm for the flap. The combined chord of the wing, which joins the leading edge of the main wing and the trailing edge of the flap is 380 mm. The span of the wing is 1100 mm, and the dimensions of the endplates are 400 x 170 x 4 mm. The optimization studies conducted by Zerihan [2001] showed that a gap of $0.032c$ and an overlap of $0.024c$ is the most efficient location for the flap. The angle of attack for the main wing and flap are 3.45° and 22.39° , respectively. This gives a true angle of attack of 14.6° for the wing and a reference angle of attack of 0° . The distance between the ground and the lowest point of the wing with a 0° reference angle of attack is defined as ground clearance.

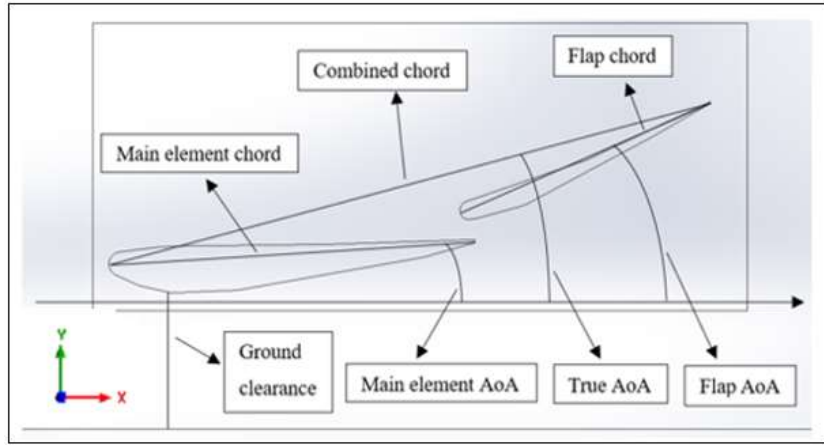


Figure 3.1: Definitions of the front wing.

The validation case is done at a ground clearance of $h/c = 0.211$ and an angle of attack of 14.6° based on the experiments of Zerihan [2001]. In the second phase of the study, the angle of attack of the wing is increased to 22° by rotating the wing about the quarter-chord point of the main wing. In the last phase of the study, rectangular CtSVGs with $h/\delta \approx 0.4$, where h is the vane height and δ is the boundary layer thickness, are added along the wing span of the main wing. Since h/δ is between 10% and 50%, the vortex generators used in this study can be identified as SVGs (Lin, 2002). The vane height of the CtSVGs is 6 mm, and the boundary layer thickness is measured at $h/c = 0.315$ for two Reynolds numbers. A total of 11 CtSVGS are used in the simulations, which gives 22 CtSVGS with the symmetry boundary condition. $2z/b$ is defined as the spanwise distance in this study, where z is the distance in the spanwise direction from the mid-span, and b is the span length. The symmetry plane ($2z/b = 0$) is at the middle of two CtSVG pairs. Vane length, the separation of the vanes at the trailing edge and the distance between two CtSVG pairs is 24 mm. The angle between the incoming flow and the vanes is 15° . The thickness of the vanes is 0.6 mm, and the leading edge of the CtSVGs is at 0.284 chord of the main wing. The size and the shape of CtSVGs are based on the literature, which can be found in Figure 3.2 [Kuya et al., 2009].

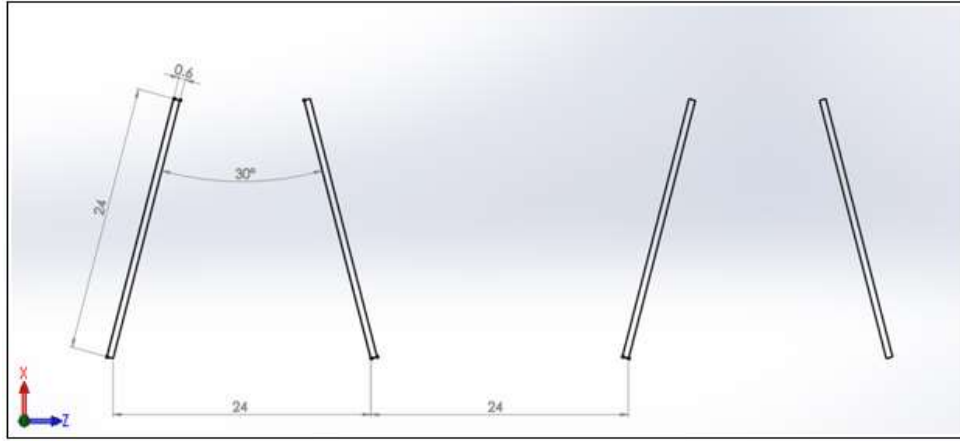


Figure 3.2: Dimensions of the CtSVGs in millimeters.

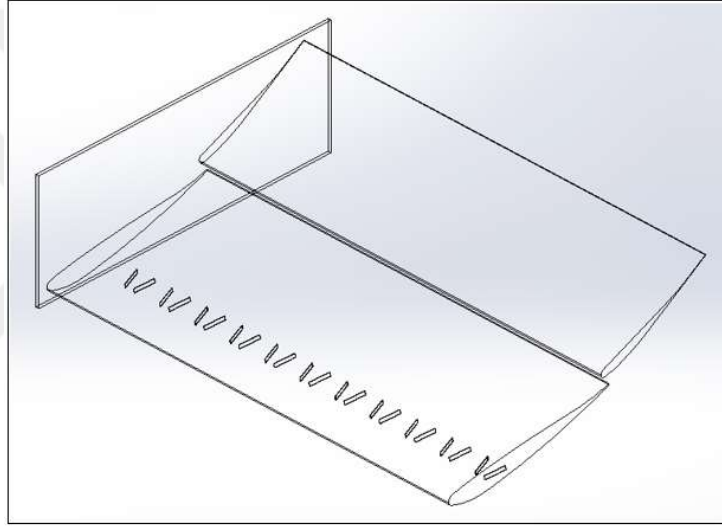


Figure 3.3: Model of the front wing with vortex generators.

3.2. Mesh and Boundary Conditions

The computational domain used for the study is a rectangular prism which has a length of $23c$, a width of $7c$ and a height of $7c$, which corresponds to 8740 mm, 2660 mm and 2660 mm respectively. A refinement box for the wake region of the wing was used based on the literature [Arrondeau and Rana, 2020]. Two other refinement boxes were used around the wing to capture the ground effect and flow separation.

Curvature and proximity functions were used for the surface meshing of the wing, and the mesh was refined at the trailing and leading edges of the main

wing and flap. Twenty inflation layers with a growth ratio of 1.2 are applied on the main wing, flap and the ground to capture the boundary layer. y^+ is a dimensionless parameter which defines the number of grid points placed in the viscous sublayer. $y^+ \leq 1$ is targeted to resolve the boundary layer as they provide sufficient resolution. y^+ is defined as [Versteeg et al., 1995]:

$$y^+ = \frac{U_\tau y}{\nu} \quad (3.1)$$

where U_τ is the friction velocity, y is the distance from the wall and ν is the kinematic viscosity. U_τ is defined as:

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (3.2)$$

where τ_w is the shear stress at the wall and ρ is the density.

A pressure outlet and a velocity inlet were used for the boundary conditions. The no-slip condition was chosen for the wing surface, endplates and ground. A moving ground which has the same velocity as the inlet velocity was used to simulate the ground effect. For the top and side boundaries, a symmetry boundary condition was used.

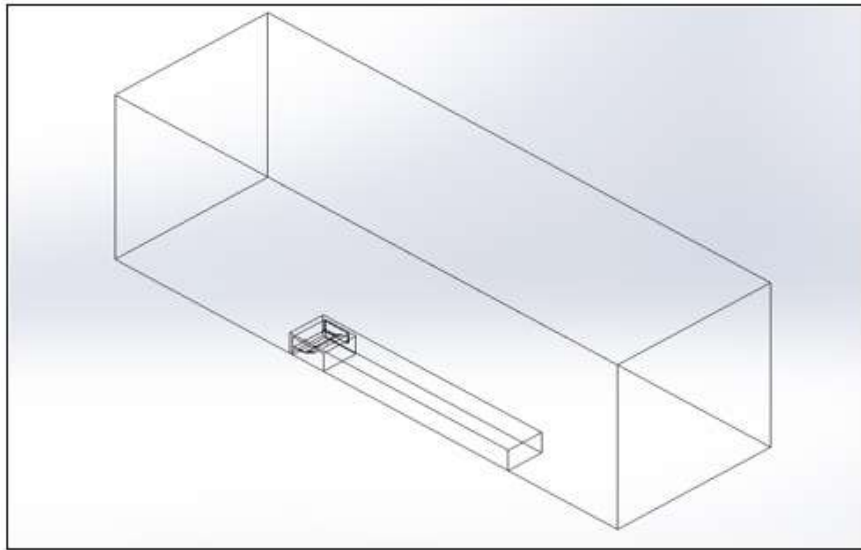


Figure 3.4: Domain and the refinement boxes.

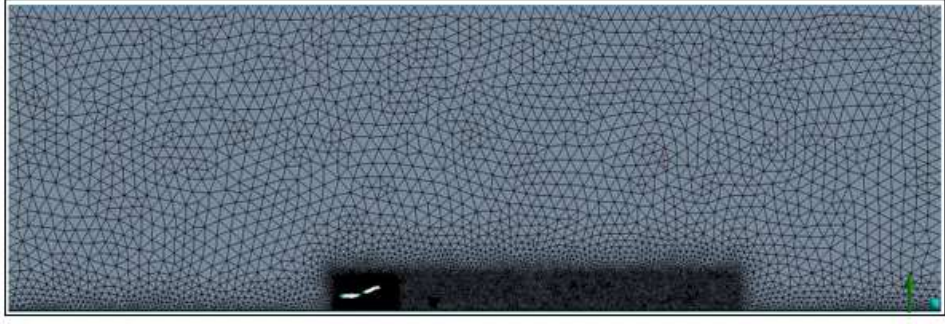


Figure 3.5: Unstructured grid used for the simulations.

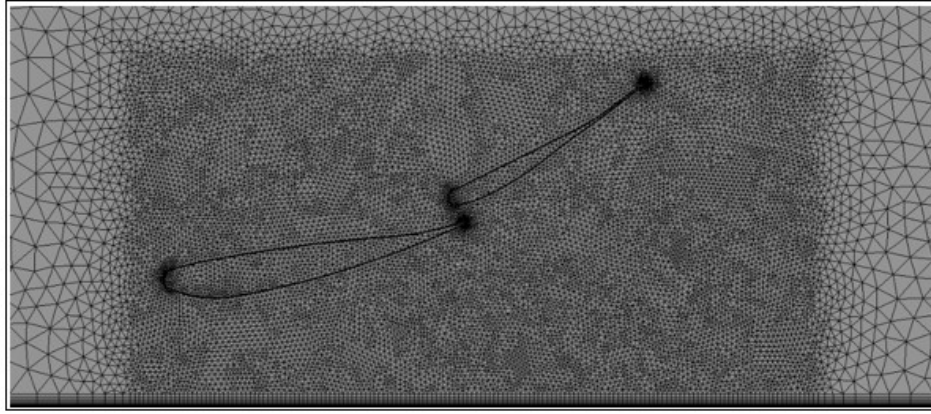


Figure 3.6: Unstructured grid used for the front wing.

3.3. Governing Equations

Computational methods were employed in this study to evaluate the performance of the wing. Three-dimensional Reynolds-averaged Navier Stokes (*RANS*) equations which are based on the conservation of mass, momentum and energy were solved by the CFD software FLUENT. The conservation of mass equation mentioned below [Çengel and Cimbala, 2018] is solved with iterative methods.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \quad (3.3)$$

Steady-state and incompressible flow assumptions are made in this study, and therefore mass conservation equation becomes,

$$\nabla \cdot (\rho V) = 0 \quad (3.4)$$

The conservation of momentum equations in Cartesian coordinates are shown as (Anderson, 2007),

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + S_x + (F_x)_{viscous} \quad (3.5)$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + S_y + (F_y)_{viscous} \quad (3.6)$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + S_z + (F_z)_{viscous} \quad (3.7)$$

Those equations are also called Navier-Stokes equations. The first and second term represent inertia forces, the third term represents pressure forces, the fourth term represents body forces and the last term represents viscous forces. The viscous forces include τ (*tau*) variables which are the components of the stress tensor. Stress tensor terms can be approximated by turbulence models. The flow was considered steady, viscid and incompressible in this study, and body forces were neglected. The Navier-Stokes equations used in this study can be written as,

$$\nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + (F_x)_{viscous} \quad (3.8)$$

$$\nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + (F_y)_{viscous} \quad (3.9)$$

$$\nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + (F_z)_{viscous} \quad (3.10)$$

Continuity and conservation of momentum equations are sufficient for incompressible flows because the main variables for incompressible flows are pressure and velocity. Density is not constant for compressible flows, and

therefore the conservation of energy equation is necessary to solve the additional variable ρ . The conservation of energy equation is not solved in this study due to the incompressible flow assumption. The conservation of energy equation can be written as [Anderson, 2007],

$$\begin{aligned} \frac{\partial}{\partial t} \left[\rho \left(e + \frac{V^2}{2} \right) \right] + \nabla \cdot \left[\rho \left(e + \frac{V^2}{2} \right) V \right] \\ = \rho \dot{q} - \nabla \cdot (pV) + \rho (f \cdot V) + \dot{Q}'_{viscous} + \dot{W}'_{viscous} \end{aligned} \quad (3.11)$$

Turbulence models are required to solve the effects of turbulence. Most CFD packages use RANS turbulence models to predict flow behavior. RANS equations can be written as [Tennekes and Lumley, 2020],

$$\begin{aligned} \rho \left(\frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} \right) \\ = F_x - \frac{\partial \bar{p}}{\partial x} + \mu \Delta \bar{u} - \rho \left(\frac{\partial \overline{u'u'}}{\partial x} + \frac{\partial \overline{u'v'}}{\partial y} + \frac{\partial \overline{u'w'}}{\partial z} \right) \end{aligned} \quad (3.12)$$

$$\begin{aligned} \rho \left(\frac{\partial \bar{v}}{\partial t} + \bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{v}}{\partial z} \right) \\ = F_y - \frac{\partial \bar{p}}{\partial y} + \mu \Delta \bar{v} - \rho \left(\frac{\partial \overline{uv'}}{\partial x} + \frac{\partial \overline{v'v'}}{\partial y} + \frac{\partial \overline{v'w'}}{\partial z} \right) \end{aligned} \quad (3.13)$$

$$\begin{aligned} \rho \left(\frac{\partial \bar{w}}{\partial t} + \bar{u} \frac{\partial \bar{w}}{\partial x} + \bar{v} \frac{\partial \bar{w}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} \right) \\ = F_z - \frac{\partial \bar{p}}{\partial z} + \mu \Delta \bar{w} - \rho \left(\frac{\partial \overline{u'w'}}{\partial x} + \frac{\partial \overline{v'w'}}{\partial y} + \frac{\partial \overline{w'w'}}{\partial z} \right) \end{aligned} \quad (3.14)$$

RANS turbulent models are used to solve the flow properties and Reynolds stresses. The literature review shows the effectiveness of different turbulence models in predicting the flow behavior over a front wing in the ground effect. Mahon and Zhang [2005] compared six turbulence models which are the standard k- ϵ [Launder and Spalding, 1974], the Realizable k- ϵ [Shih et al., 1995], the k- ϵ RNG [Yakhot and Orszag, 1986], the standard k- ω [Wilcox, 1986], the k- ω SST

[Menter, 1994] and Spalart-Allmaras [Spalart et al., 1992]. The most accurate surface pressure distribution was obtained with the $k-\omega$ SST turbulence model, whereas the most accurate wake flow field predictions were obtained with the Realizable $k-\varepsilon$ turbulence model. Accurate predictions of surface pressure distribution are required in order to obtain the downforce and drag generation of the front wing, and therefore the $k-\omega$ SST turbulence model is used in this study.

3.4. Parameters

The front wing is significantly affected by ground clearance and Reynolds number. Therefore, in the first phase of the study, the front wing's behavior at various ground clearances and Reynolds numbers was examined. The investigated flow velocities are 30 m/s and 14.4 m/s for all cases, which correspond to typical cornering and hard break conditions, respectively. The front wing has an AoA of 14.6° in this phase.

Furthermore, the angle of attack of the front wing is increased to 22° in the second phase of the study, and the effects of the varying ground clearance and Reynolds number on the wing performance are again investigated. Since the angle of attack is increased, flow separation occurs depending on the ground clearance and Reynolds number. The change of the flow separation locations with Reynolds number and ground clearance determines the location of the VGs. Clean wing configuration is used in the phase 1 and phase 2 cases, meaning CtSVGs are not used on the geometry.

CtSVG wing configuration is used in the last phase of the study, which shows the effectiveness of VGs in terms of separation control with varying Reynolds number and ground clearance. The VG setup is not changed throughout those simulations. The AoA of the front wing is 22° in the second and third phase cases. The parameters used in the analyses are listed in Table 3.1:

Table 3.1: The list of parameters used for the first, second and third phase cases.

	Ground Clearance [-]	Reynolds Number [-]	Flow Velocity [m/s]	Flow Velocity [km/h]	AoA [Degrees]	VG	Phase
Case 1	0,211	4.3×10^5	30	108	14,6	×	1
Case 2	0,211	2×10^5	14,4	51,8	14,6	×	1
Case 3	0,263	4.3×10^5	30	108	14,6	×	1
Case 4	0,263	2×10^5	14,4	51,8	14,6	×	1
Case 5	0,315	4.3×10^5	30	108	14,6	×	1
Case 6	0,315	2×10^5	14,4	51,8	14,6	×	1
Case 7	0,211	4.3×10^5	30	108	22	×	2
Case 8	0,211	2×10^5	14,4	51,8	22	×	2
Case 9	0,263	4.3×10^5	30	108	22	×	2
Case 10	0,263	2×10^5	14,4	51,8	22	×	2
Case 11	0,315	4.3×10^5	30	108	22	×	2
Case 12	0,315	2×10^5	14,4	51,8	22	×	2
Case 13	0,211	4.3×10^5	30	108	22	✓	3
Case 14	0,211	2×10^5	14,4	51,8	22	✓	3
Case 15	0,263	4.3×10^5	30	108	22	✓	3
Case 16	0,263	2×10^5	14,4	51,8	22	✓	3
Case 17	0,315	4.3×10^5	30	108	22	✓	3
Case 18	0,315	2×10^5	14,4	51,8	22	✓	3

4. RESULTS AND DISCUSSION

The characteristics of the pressure distribution, flow separation and the effect of vortex generators on the front wing are investigated in this section. A base case is presented to validate the results of the analyses and a mesh independence study is done to confirm the validity of the results.

4.1. Mesh Independence and Validation

Mesh independence study is done with five levels of grids. The number of elements used for the coarse 1, coarse 2, medium, fine and fine 2 mesh are 10M, 18.2M, 22M, 26,7M and 30M, respectively. The lift coefficient and the drag coefficient were used as parameters for the calculations. The variation in C_L and C_D decreases as the number of elements increase as shown in Figure 4.1 and Figure 4.2, therefore the results are converged. The medium mesh was chosen for the optimum computational time and stability. The case used for the validation of the results is based on the study of Zerihan [2001]. The ground clearance (h/c) is 0.211, and inlet velocity is 30 m/s for the validation case.

Table 4.1: Comparison of the C_D and C_L with the experimental results.

Coefficient	Experimental	Coarse Mesh 1	Coarse Mesh 2	Medium Mesh	Fine Mesh	Fine Mesh 2
C_D	0.254	0.24	0.259	0.262	0.263	0.264
C_L	-2.6	-1.9	-2.35	-2.39	-2.4	-2.41

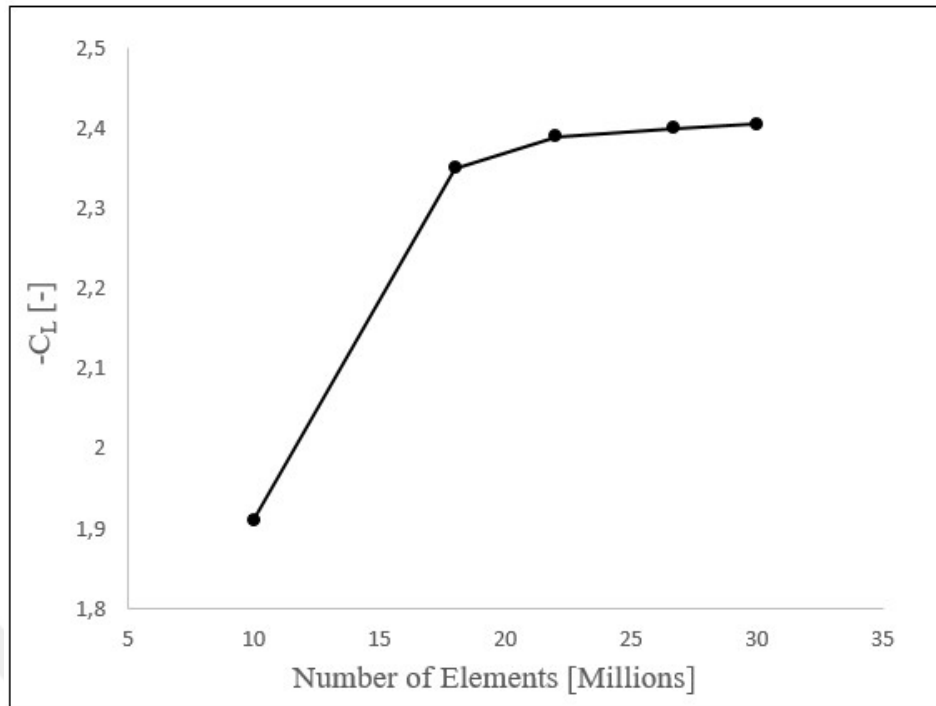


Figure 4.1: Graph of the number of elements against the downforce coefficient.

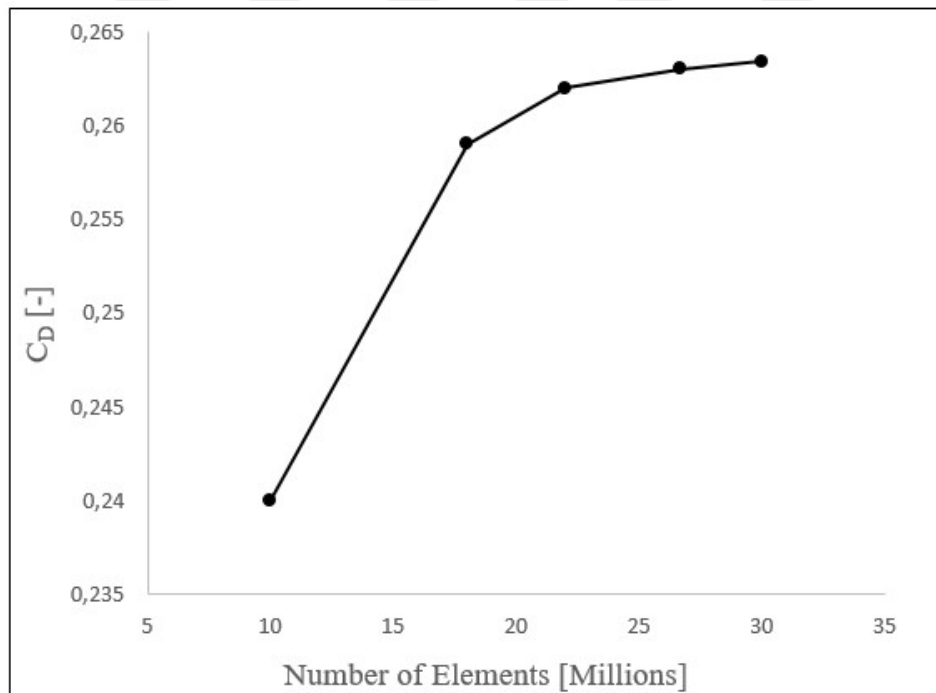


Figure 4.2: Graph of the number of elements against the drag coefficient.

The results in Table 4.1 show that the errors using the medium mesh for the lift and drag are -3,15% and 8,08%, respectively. The pressure distribution of the

wing is shown in Figure 4.3. Characteristics of the pressure distribution results are in agreement with the literature [Zerihan, 2001], however pressure coefficient (C_p) on the suction surface is mostly over-predicted by 10%. It can be seen that the results are converged and capture the general flow behavior. The reason for the differences is the limitations of the k- ω SST turbulence model. Those differences are within an acceptable range, and therefore it is reasonable to proceed with the other analyses using this setup.

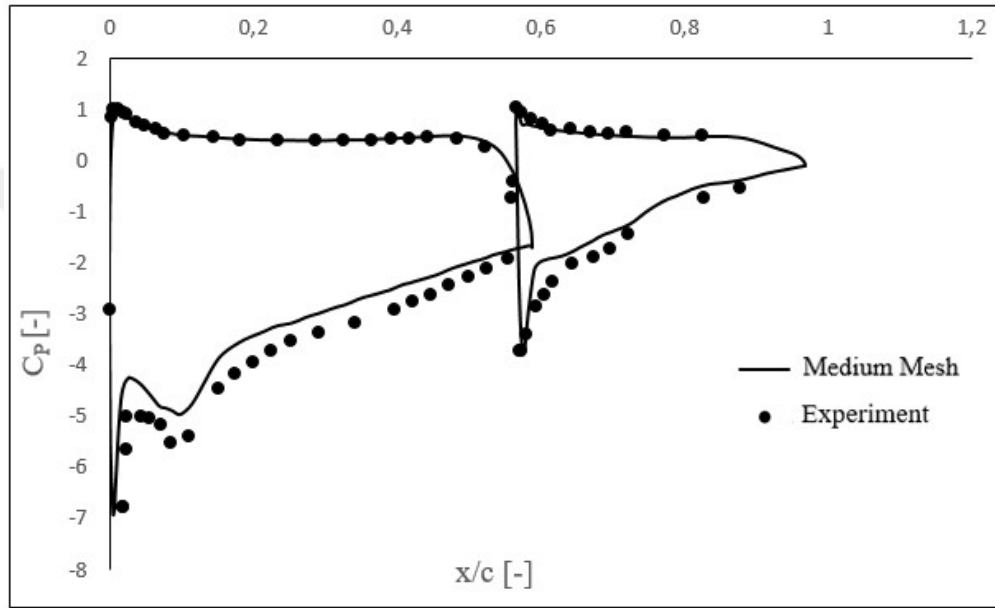


Figure 4.3: Pressure coefficient values at the symmetry plane.

4.2. Clean Wing at Low AoA (Phase 1 Cases)

This section includes simulations which are conducted at $\text{AoA} = 14.6^\circ$. The effects of the force reversal phenomenon are not observed in this section, because the critical ground clearance is not reached. The flow separation does not occur within the range of the chosen ground clearance and Reynolds number.

The results of Phase 1 cases (Figure 4.4) show that the downforce coefficient increases as the ground clearance decreases. The flow is constrained by the ground under the wing, and therefore the flow goes faster. Reducing the ground clearance strengthens this effect, and the flow is accelerated even further. The pressure under the wing decreases due to this accelerated flow, according to

the Bernoulli's principle. The pressure difference between the suction and pressure sides increases due to the accelerated flow by the reduced ground clearance, and more downforce is generated. The same behavior is observed regardless of the Reynolds number.

Figure 4.5 shows that the drag coefficient increases as the ground clearance decreases. The average increase in drag coefficient is about 4% within the studied ground clearances.

Figure 4.4 and Figure 4.5 show the effect of the Reynolds number on drag and downforce coefficients. The Reynolds number dependency is small in the force-enhancement region, and the results are consistent with the literature [Roberts et al., 2016]. The average increase in C_L and C_D with Reynolds number are 1.46% and 0.38%, respectively.

Figure 4.6 and Figure 4.7 shows the pressure distribution on the main wing and flap at the symmetry plane. In general, for all given conditions, the pressure distribution remains the same at the pressure side of the wing. However, the distribution alters for the suction side. It can be noted that decreasing the ground clearance results in higher downforce generation due to the increased pressure difference between the suction and pressure side. Ground clearance has a stronger effect than the Reynolds number on the pressure distribution of the main wing and flap, and therefore the dependency of the pressure distribution on the Reynolds number is small.

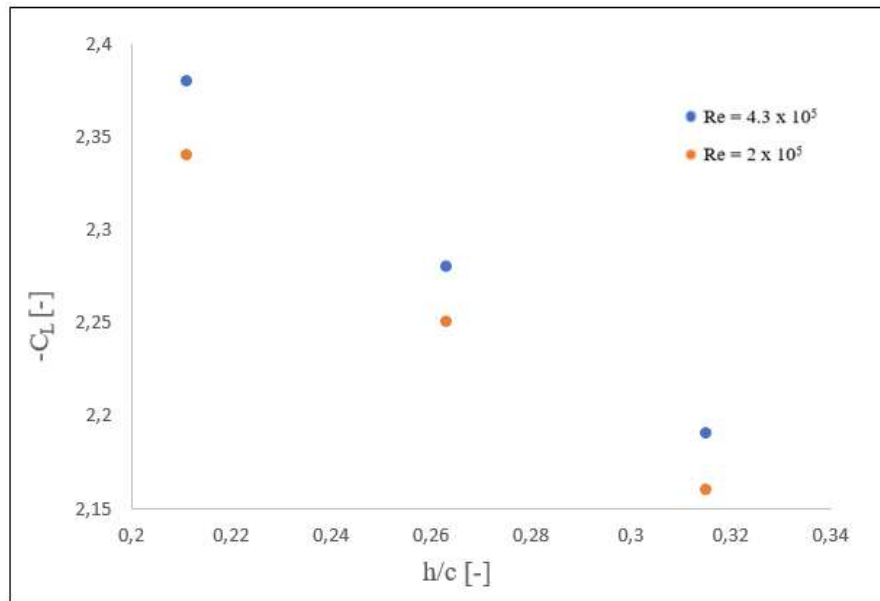


Figure 4.4: Change of downforce with ground clearance.

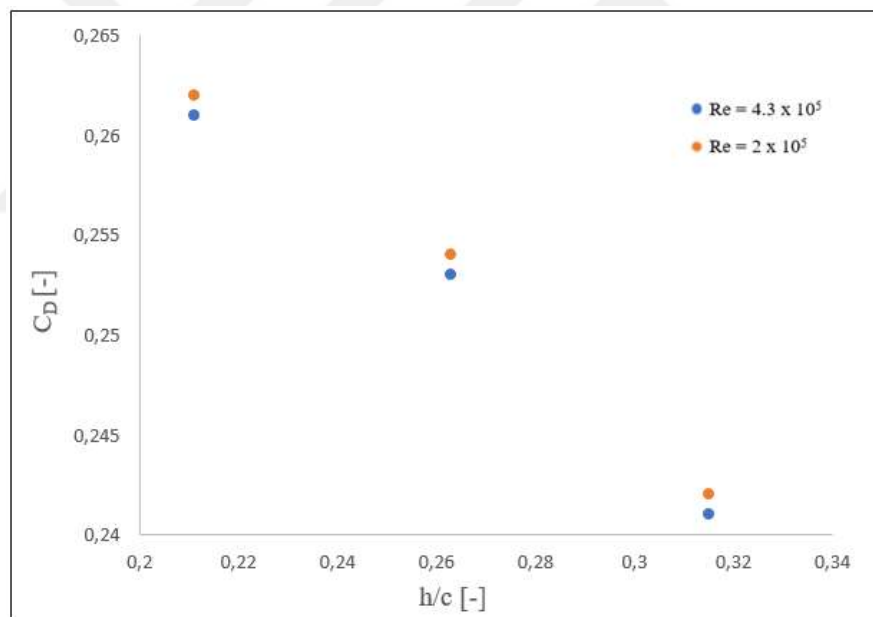


Figure 4.5: Change of drag with ground clearance.

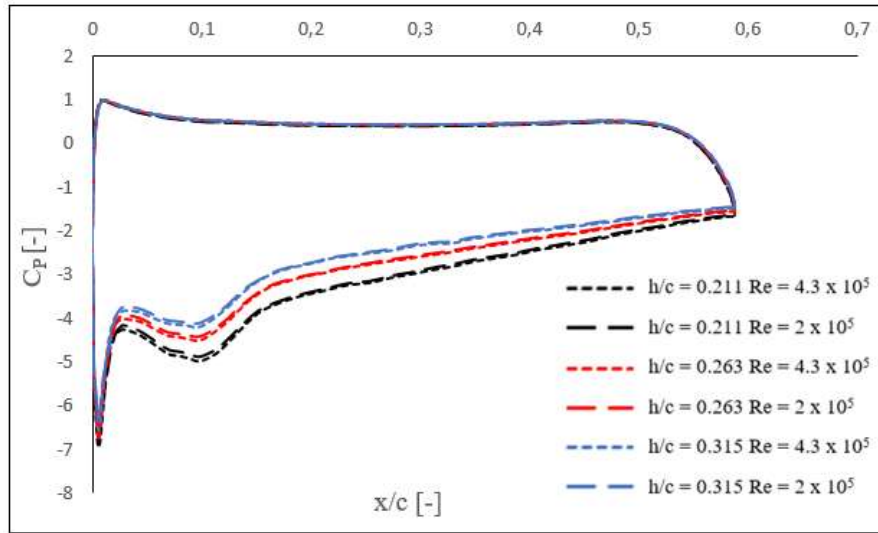


Figure 4.6: C_p of main wing at $2z/b = 0$ and $AoA = 14.6^\circ$.

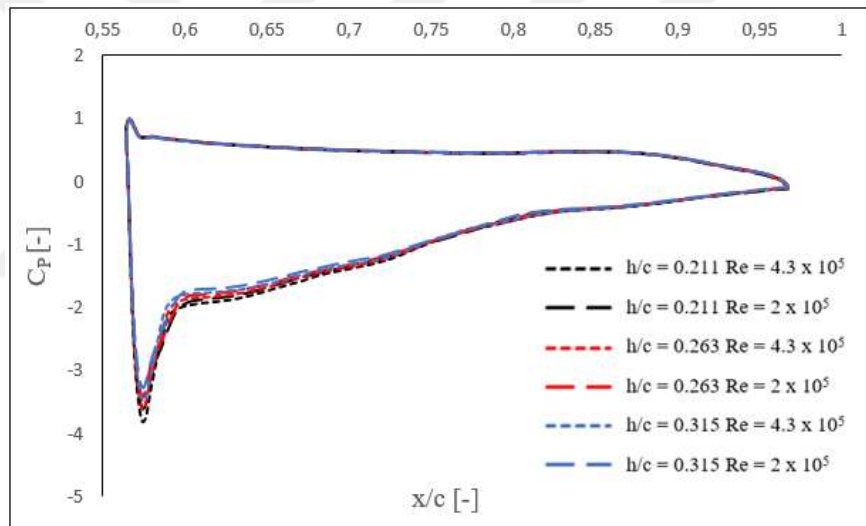


Figure 4.7: C_p of flap at $2z/b = 0$ and $AoA = 14.6^\circ$.

Contours of velocity in the x-direction are given in Figure 4.8. As mentioned, decreasing ground clearance constrains and accelerates the flow under the wing. Figure 4.9 shows the effect of the spanwise distance ($2z/b$) on the velocity distribution. The velocity in the x-direction decreases on the suction side of the main wing as the spanwise distance increases. It can also be noted that the wake region of the wing at $2z/b = 0.92$ increases as the ground clearance decreases. Those mentioned flow behaviors do not change with the Reynolds number for the phase 1 cases.

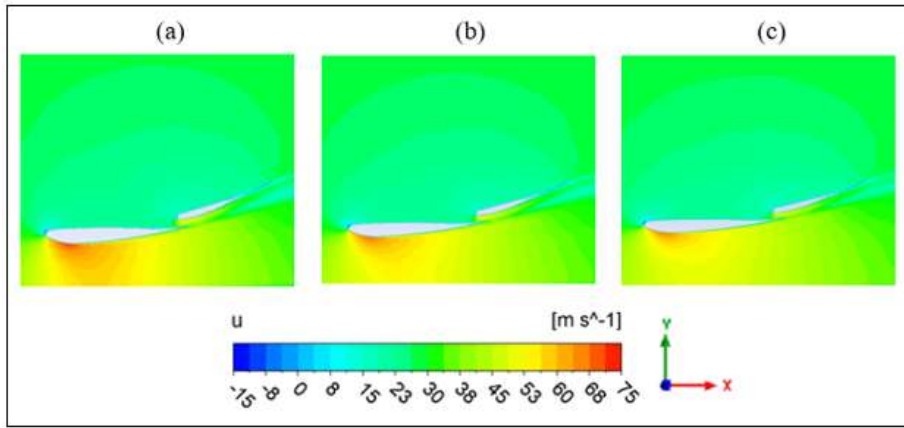


Figure 4.8: Contours of velocity in x-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

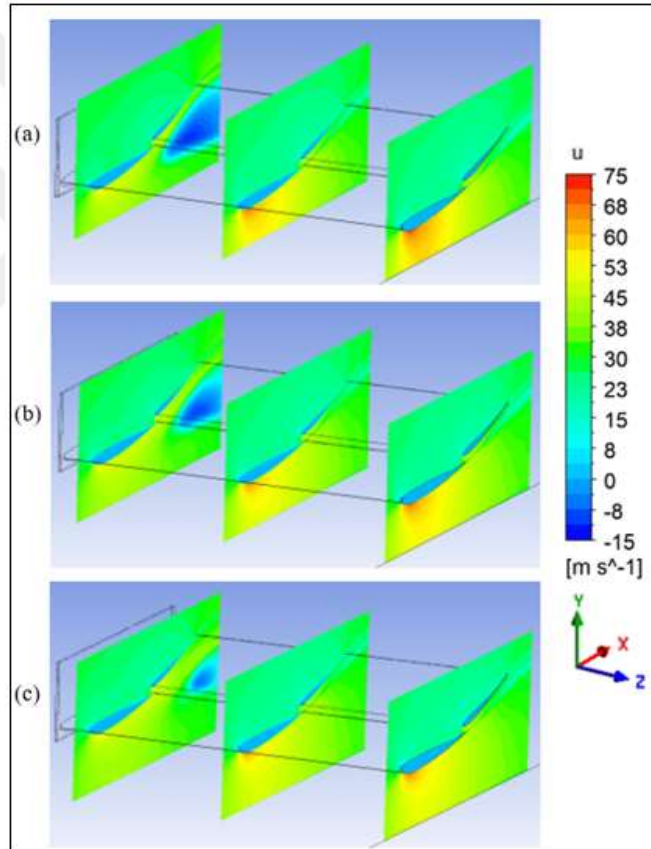


Figure 4.9: Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$ and $2z/b = 0.92$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

Vorticity contours provide reliable information about the flow separation. Figure 4.10 shows the contours of vorticity in the z -direction. Observing those contours, it can be noted that there is no flow separation at the mid-span

regardless of the studied ground clearance. Thus, the front wing is in the force-enhancement region in those cases.

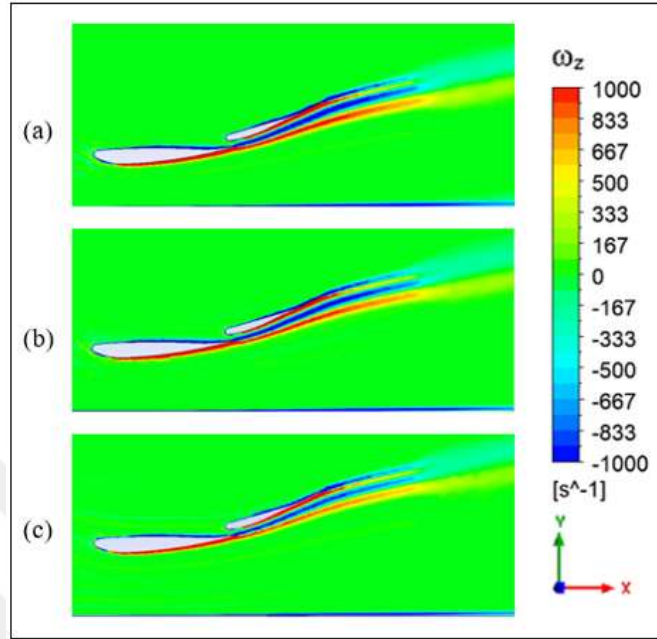


Figure 4.10: Contours of vorticity in z-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$
(a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

Another observation of those simulations is the vortices which are generated at the lower and the upper side of the endplate. Those vortices which are measured at $x/c = 1.053$ can be seen in Figure 4.11. The lower vortices proceed to break down as the ground clearance is decreased. Breakdown of the vortices may also reduce the downwash of the flow, and therefore reduce the induced drag. The strength and size of the upper vortices are not affected significantly by the change of ground clearance. This behavior of the flow does not change when the Reynolds number is decreased for the phase 1 cases.

Figure 4.12 shows the behavior of the upper and lower vortices in the wake region. The breakdown of the lower vortex occurs earlier, whereas the core of the upper vortex stays concentrated for a slightly longer distance downstream. As those two co-rotating vortices flow downstream, they merge at about $x/c = 3.289$ and then dissipate.

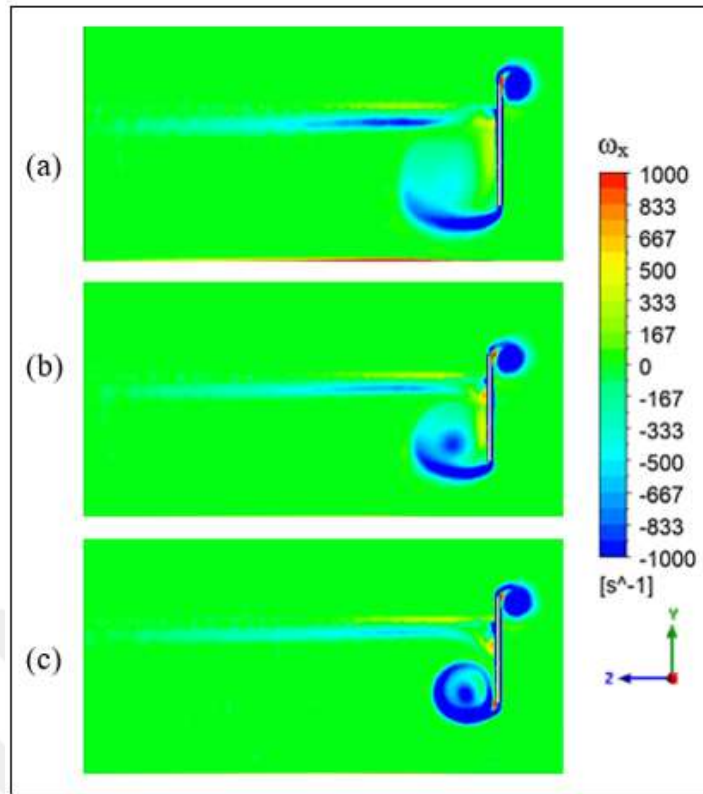


Figure 4.11: Contours of vorticity in x-direction viewed from downstream at $x/c = 1.053$ and $Re = 4.3 \times 10^5$ (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

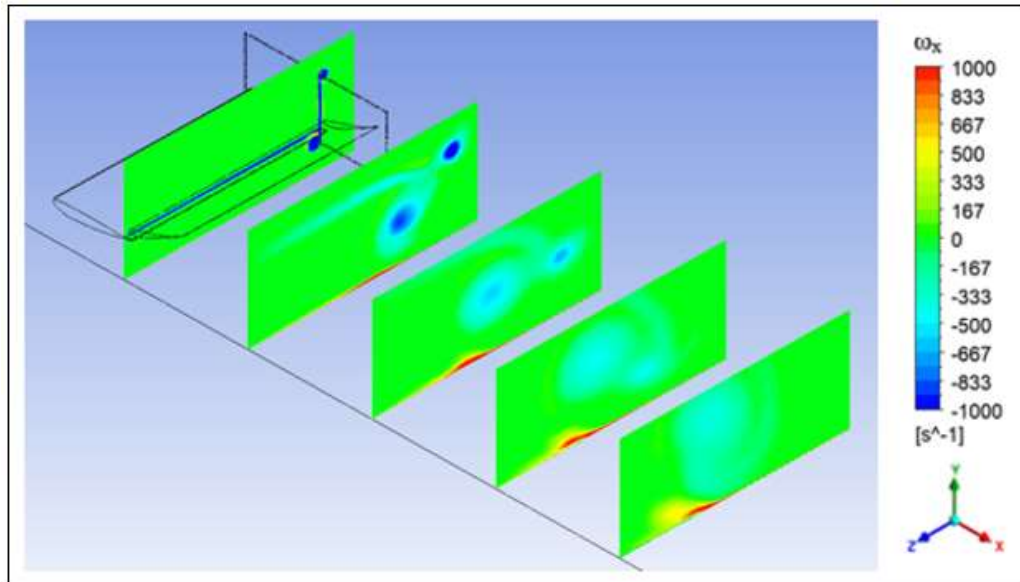


Figure 4.12: Contour of vorticity in x-direction at $x/c = 0.526$, $x/c = 1.447$, $x/c = 2.368$, $x/c = 3.289$, $x/c = 4.21$ and $Re = 4.3 \times 10^5$.

4.3. Clean Wing at High AoA (Phase 2 Cases)

As mentioned before, the critical ground clearance is reached in cases given in this section, and the flow reversal phenomenon can be observed due to the increased AoA. This flow behavior was also reported in the literature [Zerihan, 2001]. Flow separation occurs at the trailing edge of the main wing within the range of the chosen ground clearances and Reynolds numbers.

The variation of downforce and drag coefficients with varying Reynolds number and ground clearance can be observed in Figure 4.13 and Figure 4.14, respectively. Figure 4.13 shows that the downforce coefficient increases when the ground clearance is reduced at $Re = 2 \times 10^5$; however, the cases at $Re = 4.3 \times 10^5$ exhibit the opposite behavior. The highest downforce is observed at $h/c = 0.315$ and $Re = 4.3 \times 10^5$, and the lowest downforce is observed at $h/c = 0.315$ and $Re = 2 \times 10^5$. Flow separation can be prevented by increasing the ground clearance at $Re = 4.3 \times 10^5$, as shown by the sudden increase in the downforce coefficient at $h/c = 0.315$. This flow behavior will be shown in detail in Figure 4.17. The flow separation occurs in every phase 2 case at $Re = 2 \times 10^5$, which indicates that flow separation is not prevented by the increase in ground clearance. The reason for this behavior may be the increase in the critical ground clearance as the Reynolds number decreases.

Pressure distribution on the main wing can be found in Figure 4.15. In general, for all given conditions, the pressure distribution remains the same at the pressure side of the wing except the case at $h/c = 0.315$ and $Re = 4.3 \times 10^5$. However, the distribution changes for the suction side. Flow separations can be observed by the constant C_p region which is approximately between $x/c = 0.35$ and $x/c = 0.58$. Flow separation does not occur at $h/c = 0.315$ and $Re = 4.3 \times 10^5$; therefore, the constant C_p region is not observed for this case. Reynolds number has a significant effect on the pressure distribution on the main wing. C_p difference between the pressure and suction surfaces of the main wing increases as the Reynolds number is increased. Reynolds number has its highest impact at $h/c = 0.315$, which will be shown in detail in Figure 4.18. Reynolds number is a dominant factor on the flow separation at $h/c = 0.315$, and therefore, it changes the pressure distribution drastically.

Similarly, the pressure distribution on the flap can be found in Figure 4.16. The absence of the flow separation at $h/c = 0.315$ and $Re = 4.3 \times 10^5$ changes the pressure distribution of the pressure and suction surfaces of the flap. This behavior does not apply for the main wing. Only the suction surface of the main wing is affected significantly by the absence of flow separation, as can be seen in Figure 4.15.

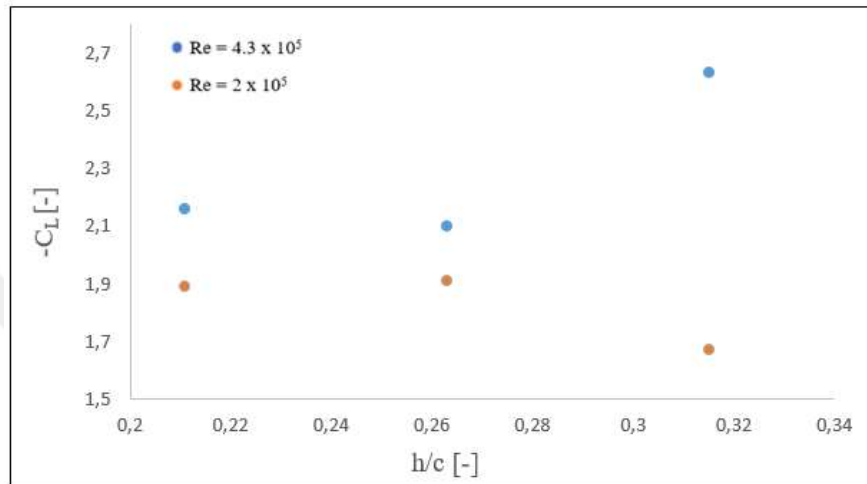


Figure 4.13: Change of downforce with ground clearance.

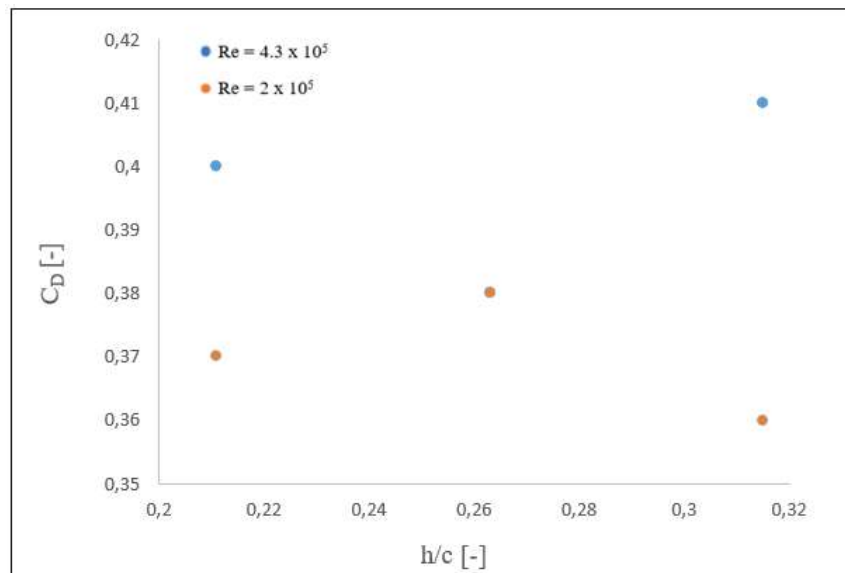


Figure 4.14: Change of drag with ground clearance.

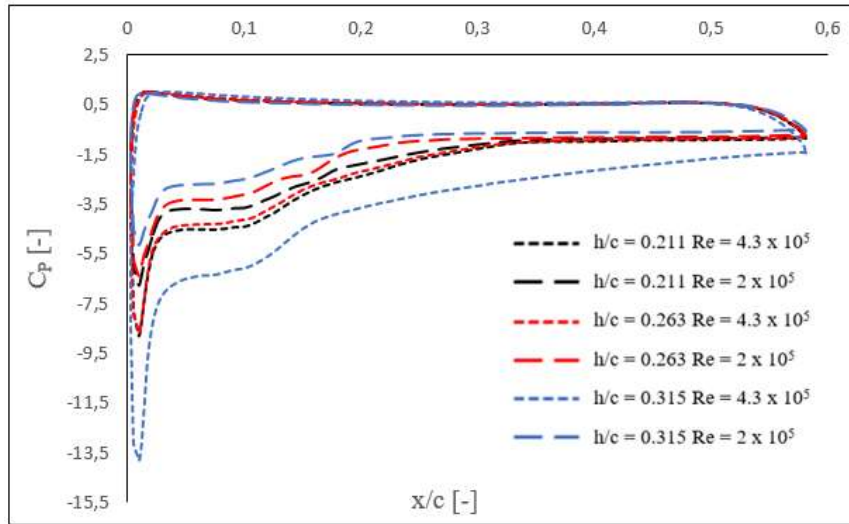


Figure 4.15: C_p of main wing at $2z/b = 0$ and $AoA = 22^\circ$.

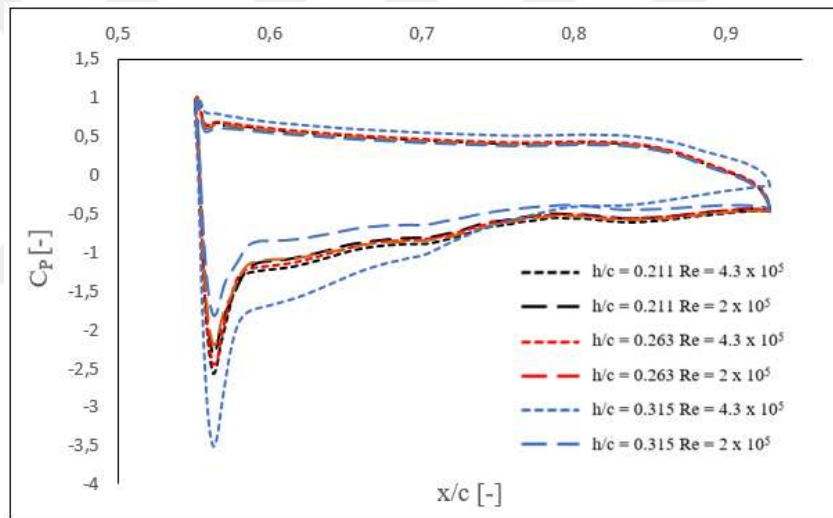


Figure 4.16: C_p of flap at $2z/b = 0$ and $AoA = 22^\circ$.

Contours of velocity in the x-direction are given in Figure 4.17. The flow separations at the mid-span can be observed in the figure. Even though the highest ground clearance within the range of cases is $h/c = 0.315$, the velocity in the x-direction under the main wing is the highest at $h/c = 0.315$ due to the flow separations occurring at lower ground clearances. Although, restricting the flow area by decreasing the ground clearance increases the flow velocity under the main wing, flow separation serves as a barrier against the flow, and decreases the flow velocity under the main wing at lower ground clearances. The opposite of

this flow behavior can be observed in Figure 4.8, where the velocity in the x-direction under the main wing increases as the ground clearance decreases. Flow separation is not observed at $h/c = 0.315$. $h/c = 0.315$ being higher than the critical ground clearance may be the reason why flow separation is not observed in this case.

The impact of the Reynolds number on the boundary layer separation can be observed in Figure 4.18. In Figure 4.18 (a), the flow is able to overcome the adverse pressure gradients, and stays attached to the main wing. The flow separates at about 0.5 chord as the Reynolds number decreases as seen in Figure 4.18 (b).

Contours of velocity in the x-direction at different spanwise sections can be found in Figure 4.19. The wake region behind the wing at mid-span is large due to the flow separation at $2z/b = 0$ which can be seen in Figure 4.19 (a) and Figure 4.19 (b). The size of the wake region decreases at $2z/b = 0.48$, possibly due to the endplate vortices adding momentum to the boundary layer of the main wing. This effect is more visible at $2z/b = 0.92$, as there is no flow separation on the main wing at this section. In Figure 4.19 (c), it can be observed that the wake size slightly decreases as the spanwise distance increases from $2z/b = 0$ to $2z/b = 0.48$, which again shows the impact of the endplate vortices on the boundary layer flow of the main wing. Flow separation does not occur on the flap at any section of the wing. The flow is able to overcome the adverse pressure gradients on the flap, possibly due to the constrained flow coming from the gap between the main wing and the flap.

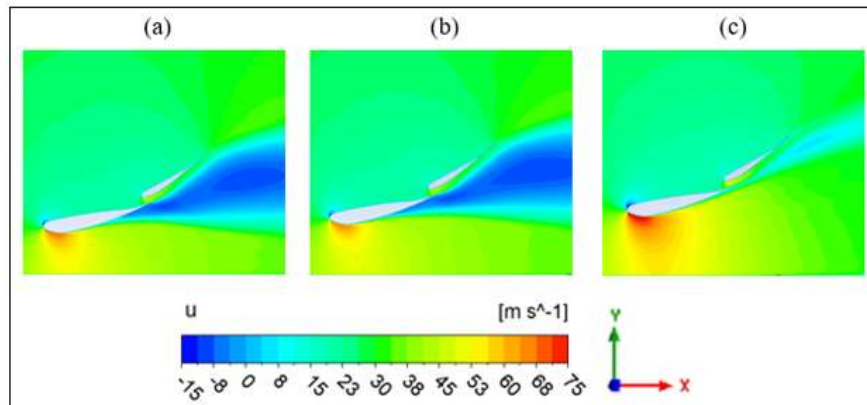


Figure 4.17: Contours of velocity in x-direction at $2z/b = 0$ and $\text{Re} = 4.3 \times 10^5$;
(a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

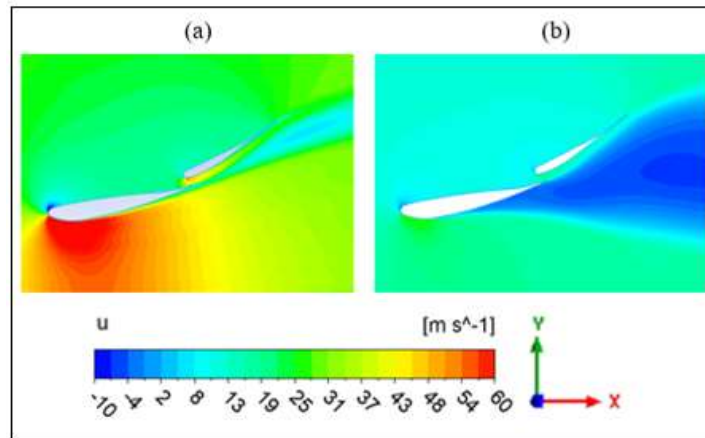


Figure 4.18: Contours of velocity in x-direction at $2z/b = 0$ and $h/c = 0.315$ (a) $Re = 4.3 \times 10^5$ (b) $Re = 2 \times 10^5$.

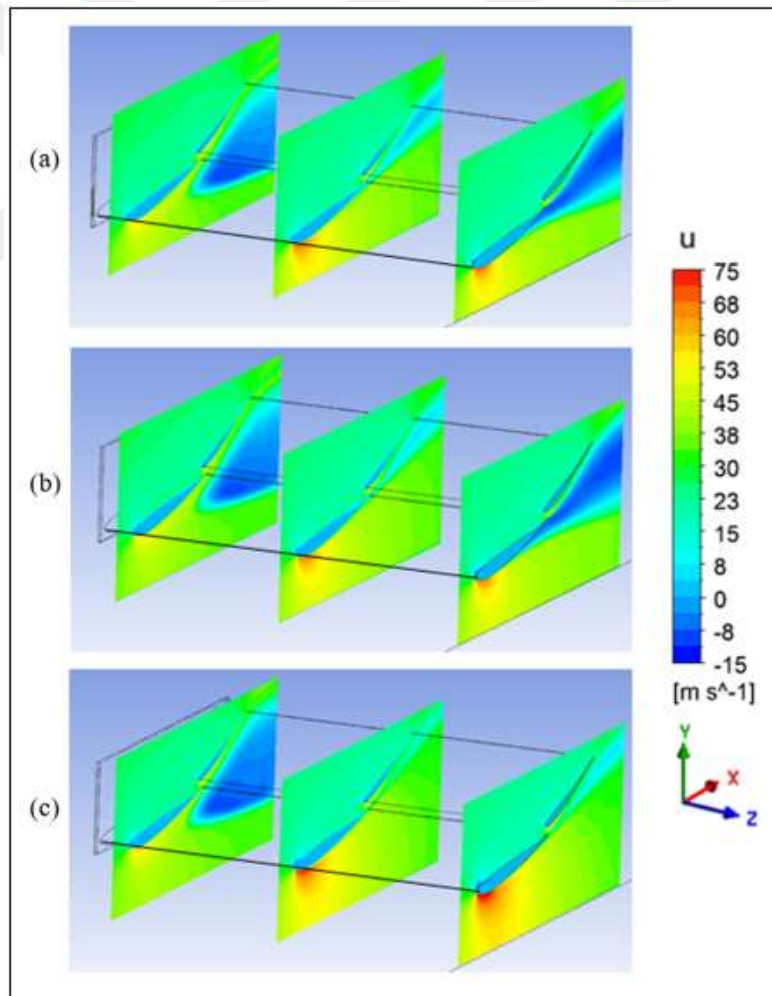


Figure 4.19: Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$, $2z/b = 0.92$ and $Re = 4.3 \times 10^5$; (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

Contours of vorticity in z-direction at different ground clearances are shown in Figure 4.20. The flow separations can be observed in Figure 4.20 (a) and Figure 4.20 (b). The size of the wake region does not change with ground clearance when there is a flow separation on the wing. Flow separation is not observed in Figure 4.20 (c) which is in line with the previous results.

In Figure 4.21, vortex breakdowns at $x/c = 1.053$ can be observed. The size of the lower vortex grows slightly as the ground clearance increases. The merging and dissipation of the lower and upper vortices can be seen in Figure 4.22. The cores of the upper and lower vortices are still concentrated at $x/c = 1.447$, and the merging of the vortices is initiated at about $x/c = 2.368$.

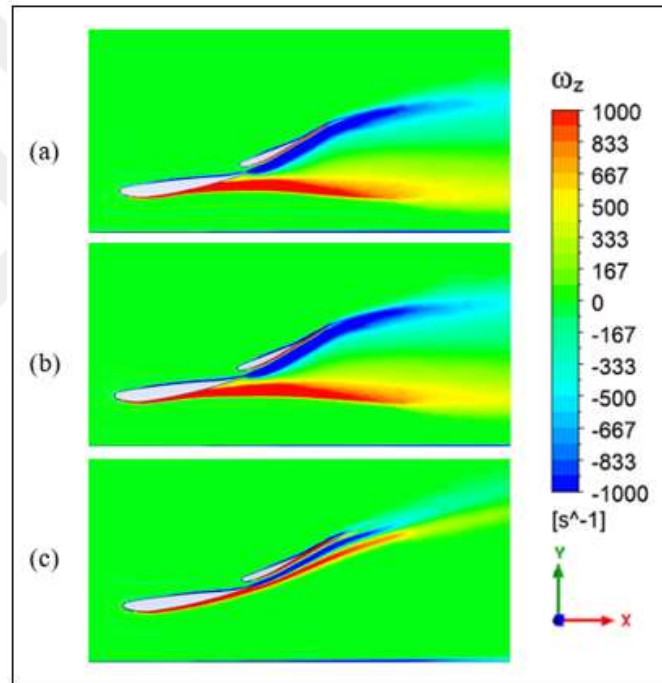


Figure 4.20: Contours of vorticity in z-direction at $2z/b = 0$ and $Re = 4.3 \times 10^5$
(a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

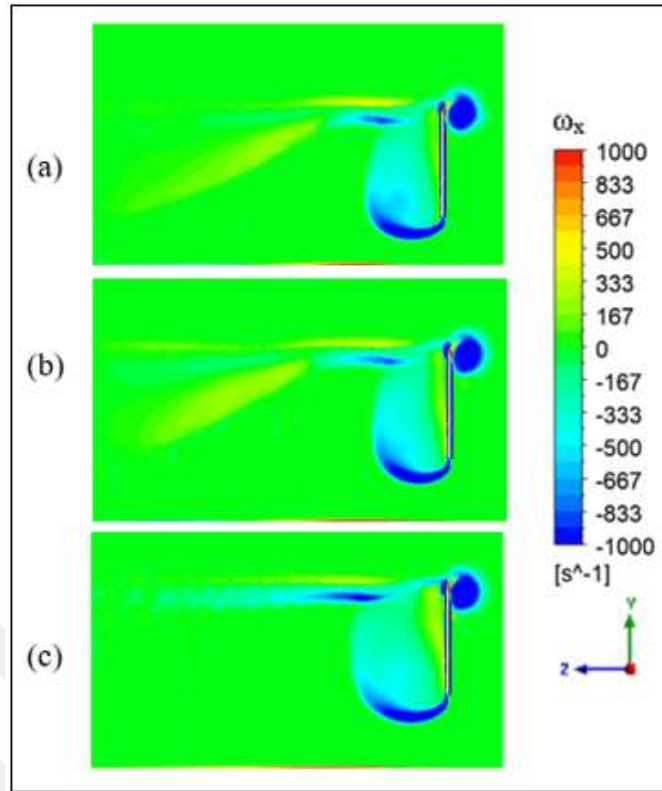


Figure 4.21: Contours of vorticity in x-direction viewed from downstream at $x/c = 1.053$ and $Re = 4.3 \times 10^5$ (a) $h/c = 0.211$ (b) $h/c = 0.263$ (c) $h/c = 0.315$.

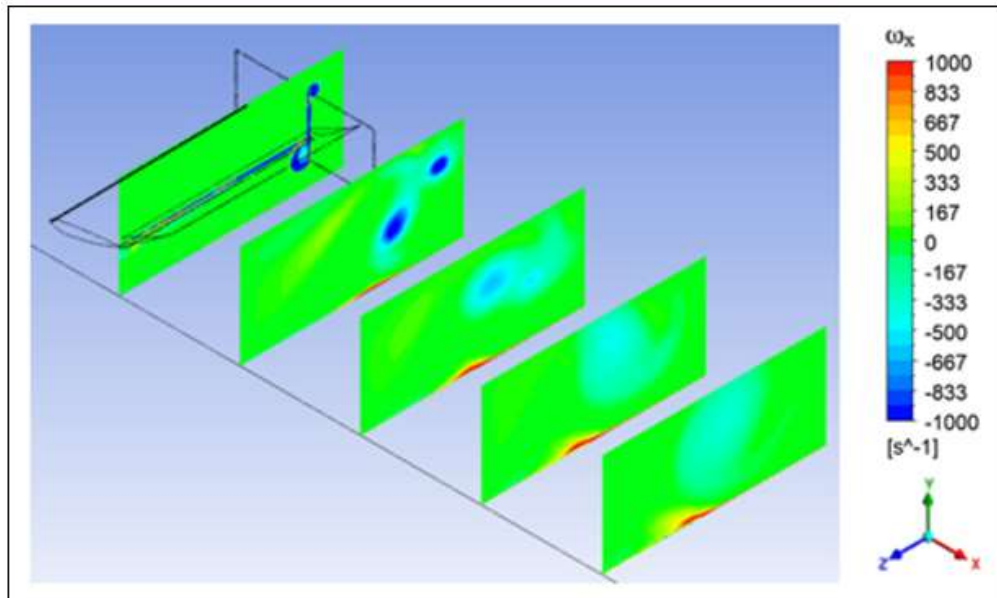


Figure 4.22: Contours of vorticity in x-direction at $x/c = 0.526$, $x/c = 1.447$, $x/c = 2.368$, $x/c = 3.289$, $x/c = 4.21$ and $Re = 4.3 \times 10^5$.

In conclusion, the numerical investigation of the flow using the parameters which are indicated for phase 1 and phase 2 cases are performed. The following conclusions for the phase 1 and phase 2 cases can be drawn:

- The flow separation is not observed in any of the phase 1 cases. Phase 2 cases exhibit flow separation except the case conducted at $h/c = 0.315$ and $Re = 2 \times 10^5$.
- Phase 2 cases provide less downforce and higher drag than phase 1 cases due to flow separations. The only exception is the case at $h/c = 0.315$ and $Re = 4.3 \times 10^5$. The wing generates more downforce and drag at this ground clearance and Reynolds number than its counterpart which is at $AoA = 14.6^\circ$.
- For the phase 1 cases, downforce increases as the ground clearance decreases at all Reynolds numbers. This behavior changes significantly for the phase 2 cases. For the phase 2 cases at $Re = 4.3 \times 10^5$, downforce slightly decreases as the ground clearance is increased from 0.211 to 0.263, and downforce increases drastically as the ground clearance is increased from 0.263 to 0.315. This behavior occurs possibly due to the force reversal phenomenon. For the phase 2 cases at $Re = 2 \times 10^5$, downforce increases as the ground clearance decreases.
- The impact of the Reynolds number on downforce and drag is small for the phase 1 cases. For the phase 2 cases, Reynolds number plays a vital role as it is a deciding factor for the occurrence of the flow separation at $h/c = 0.315$. Downforce decreases as the Reynolds number decreases for both phase 1 and phase 2 cases.
- The vortices produced by the endplate possibly increase the momentum of the boundary layer flow of the main wing for both phase 1 and phase 2 cases. The wake region of the main wing becomes smaller as the spanwise distance from the mid-span increases, possibly due to the endplate vortices.
- The lower and upper vortices at the wake region of the wing merge and dissipate for both phase 1 and phase 2 cases. However, for the phase 1 cases, the merging of the vortices occurs at about $x/c = 3.289$, whereas for the phase 2 cases, the merging occurs at about $x/c = 1.447$.

Those results provide insight into the effects of the increased AoA, along with the other parameters such as ground clearance and Reynolds number on the flow behavior.

4.4. Separation Control with CtSVGs (Phase 3 Cases)

In phase 3 cases, CtSVGs are implemented on the front wing in order to eliminate or reduce the flow separation which are observed in phase 2 cases. CtSVGs prove to be effective in eliminating or reducing the flow separations by adding momentum to the boundary layer flow. In this section, clean and CtSVG wing configurations correspond to the phase 2 and phase 3 cases, respectively.

The change of the downforce and drag coefficients with varying ground clearance can be observed in Figure 4.23 and Figure 4.24. The clean wing configuration provides less downforce at $h/c = 0.211$ and $h/c = 0.263$. The addition of CtSVGs results in an increase of downforce coefficient by 25% - 30%, and an increase of drag coefficient by 2% - 13%. The downforce coefficient increases slightly at $h/c = 0.315$, because the separation does not occur at this ground clearance with a clean wing.

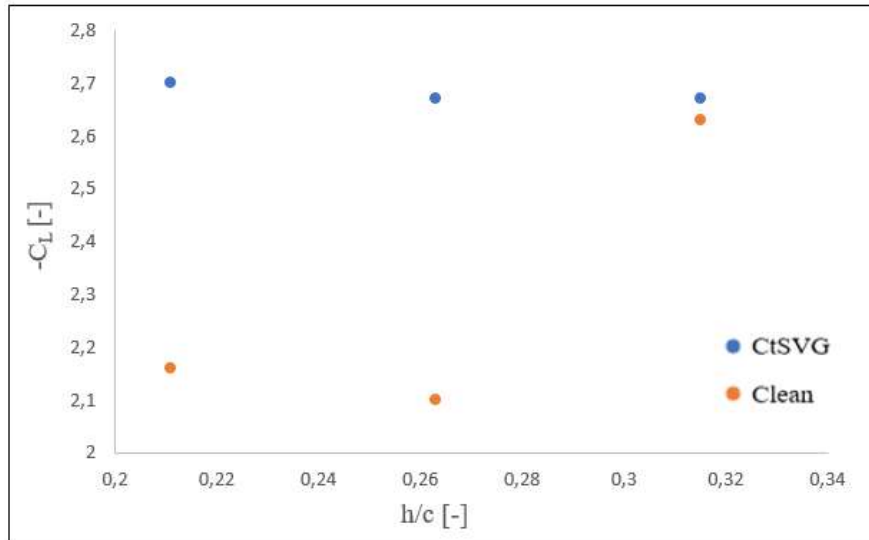


Figure 4.23: Change of downforce with ground clearance at $Re = 4.3 \times 10^5$ for clean and CtSVG wing configurations.

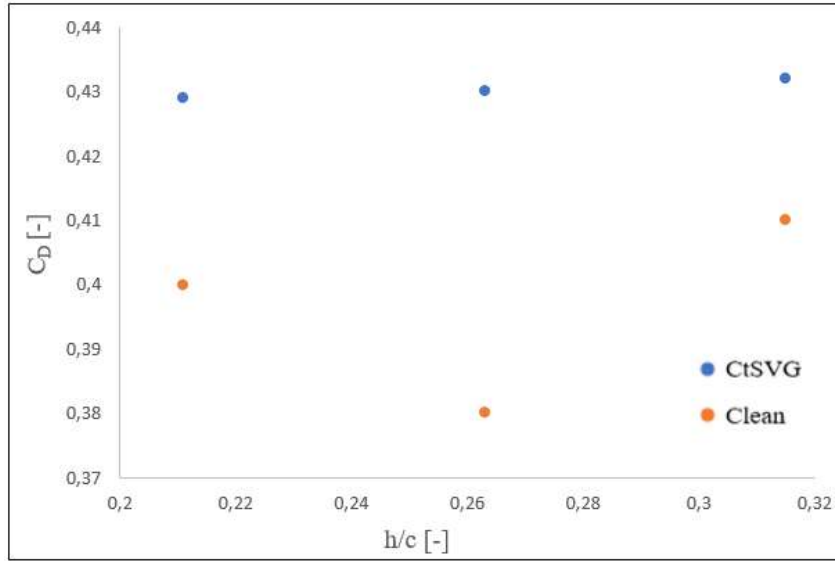


Figure 4.24: Change of drag with ground clearance at $Re = 4.3 \times 10^5$ for clean and CtSVG wing configurations.

The C_p graph of the main wing and flap at $2z/b = 0.044$ can be found in Figure 4.25 and Figure 4.26. The peak which is observed at $x/c = 0.18$ with the CtSVG configurations shows the increase in momentum at the leading edge of the vortex generators. This increase in momentum eliminates the constant C_p region, which is observed with the clean wing configurations, and therefore increases the downforce generation. The addition of the vortex generators promotes increased downforce generation on the flap as well. C_p graphs of the clean wing and the CtSVG configurations at $h/c = 0.211$ and $h/c = 0.263$ show the decrease in pressure at the suction side of the flap at the leading edge. However, the C_p graphs at the trailing edge of the flap shift upwards with the addition of vortex generators. Thus, the pressure difference between the pressure side and suction side of the flap at the leading edge increases with the addition of vortex generators, and this pressure difference does not change significantly at the trailing edge of the flap.

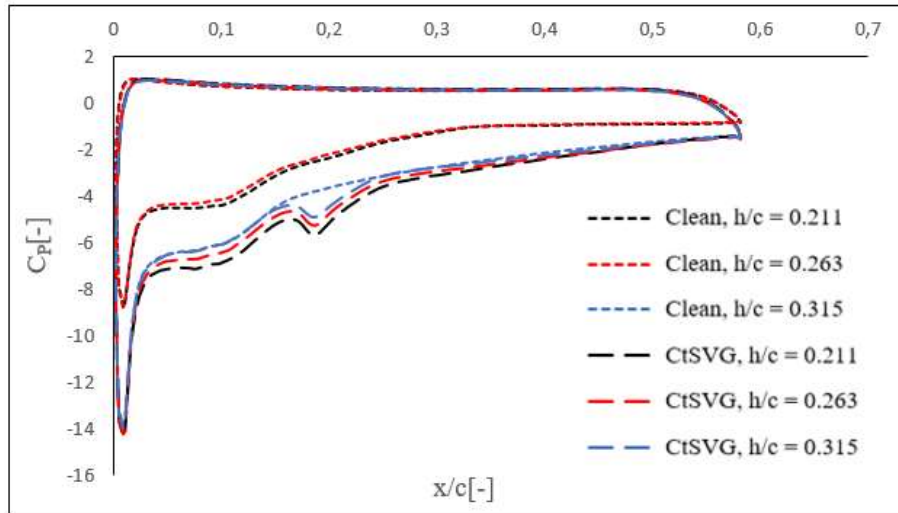


Figure 4.25: C_p of main wing at $2z/b = 0.044$, $Re = 4.3 \times 10^5$ and $AoA = 22^\circ$.

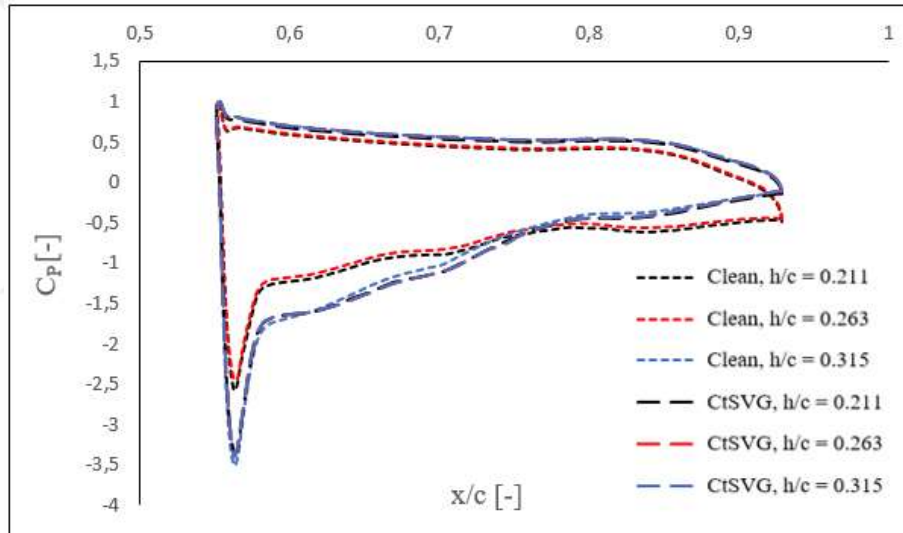


Figure 4.26: C_p of flap at $2z/b = 0.044$, $Re = 4.3 \times 10^5$ and $AoA = 22^\circ$.

Contours of velocity in the x-direction at $2z/b = 0.044$ in Figure 4.27 show the elimination of the flow separation at the trailing edge. $2z/b = 0.044$ is in the middle of the VG vanes which are at the closest spanwise location to the mid-span (i.e., symmetry plane). CtSVGs located at $x/c = 0.39$ is behind the onset of the flow separation, and therefore they are able to eliminate flow separation. The size of the wake is also reduced drastically by vortex generators.

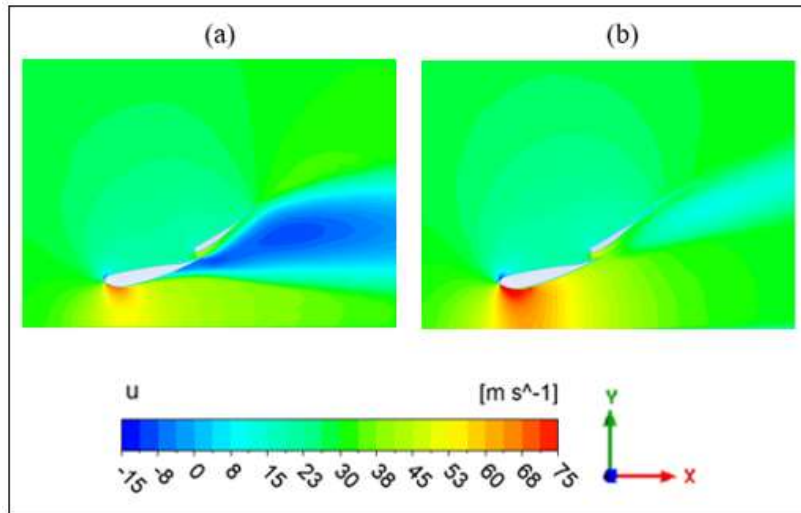


Figure 4.27: Contours of velocity in x-direction at $2z/b = 0.044$, $h/c = 0.211$ and $Re = 4.3 \times 10^5$ (a) Clean wing (b) Wing with CtSVGs.

Figure 4.28 shows the velocity distribution in different spanwise locations for clean and CtSVG wing configurations. Contours of velocity at $2z/b = 0$ show that the flow separation is reduced between the VG pairs as well. The wake size of the wing is reduced by vortex generators at $2z/b = 0.48$. However, the wake size increases slightly at $2z/b = 0.92$ with the presence of the vortex generators. Flow separation does not occur at this spanwise location on the clean wing, possibly due to the endplate vortices. Reducing the wake size at $2z/b = 0.92$ may yield less drag, and thus it may not be feasible to use vortex generators closer to the endplates. However, this requires additional research, and it is out of the scope of the current study.

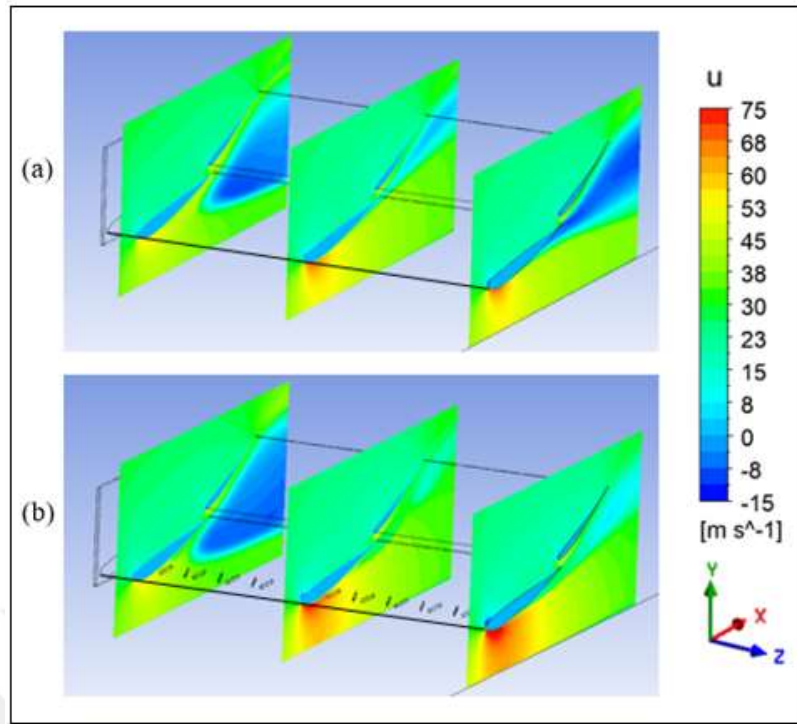


Figure 4.28: Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$, $2z/b = 0.92$ and $Re = 4.3 \times 10^5$ and $h/c = 0.211$ (a) Clean wing (b) Wing with CtSVGs.

Vortex generators prove to be effective at lower Reynolds numbers as well which can be observed in Figure 4.29 and Figure 4.30. As mentioned, flow separation does not occur on a clean wing configuration at $h/c = 0.315$ and $Re = 4.3 \times 10^5$. However, flow separation can be observed at the same ground clearance with a Reynolds number of 2×10^5 . This flow separation is prevented with the use of vortex generators as seen in Figure 4.29, and the downforce coefficients of those two cases are -1.67 and -2.58, respectively. Flow separations on a clean wing configuration at $h/c = 0.211$ and $h/c = 0.263$ could not be prevented with the use of vortex generators at $Re = 2 \times 10^5$. Those two cases using CtSVG wing configuration at $h/c = 0.211$ and $h/c = 0.263$ are not mentioned in this study due to the extreme convergence difficulties, and they exhibit unsteady characteristics even after a high number of iterations. Figure 4.30 shows the reduction of the flow separation at $2z/b = 0$ as well. The wake size is reduced at $2z/b = 0.48$ with the use of vortex generators. However, a significantly larger wake is produced at $2z/b = 0.92$ on a CtSVG configuration, which supports the idea of avoiding the use of vortex generators closer to the endplates.

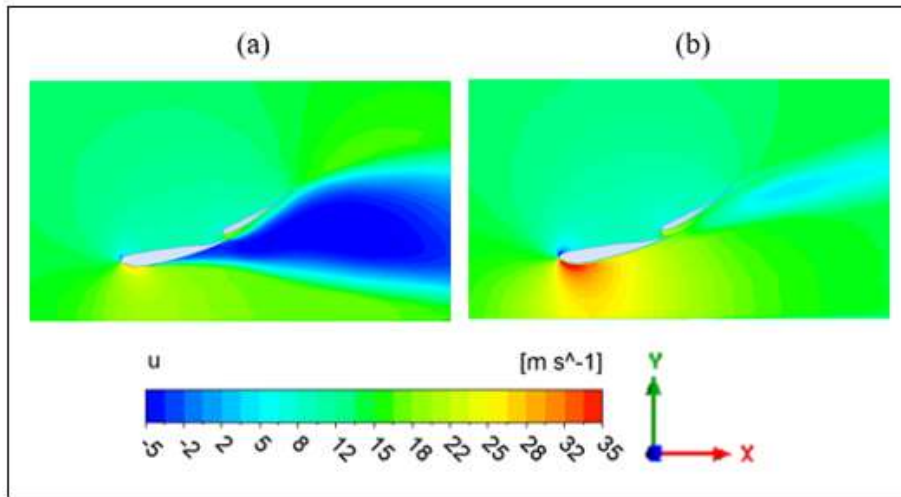


Figure 4.29: Contours of velocity in x-direction at $2z/b = 0.044$, $h/c = 0.315$ and $Re = 2 \times 10^5$ (a) Clean wing (b) Wing with CtSVGs.

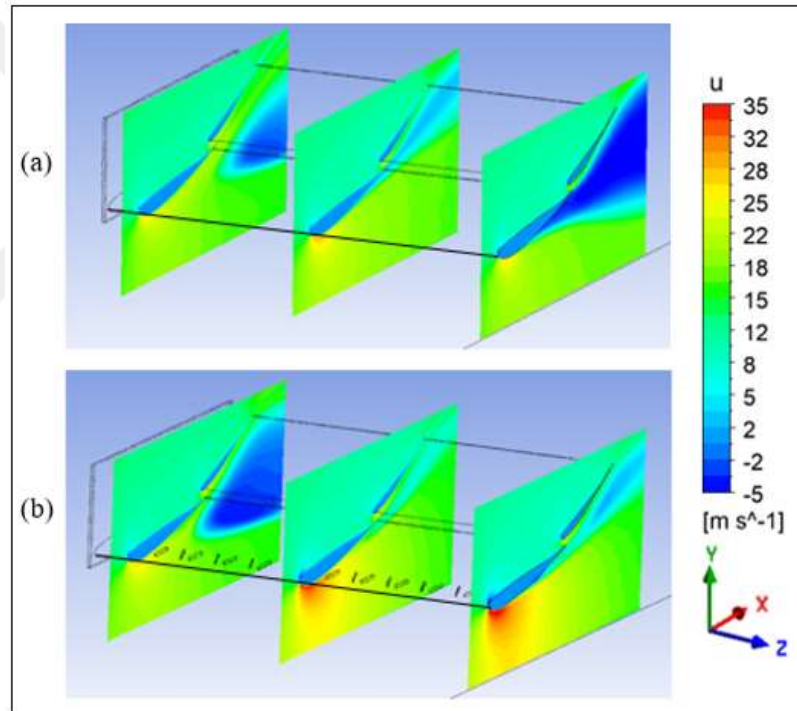


Figure 4.30: Contours of velocity in x-direction at $2z/b = 0$, $2z/b = 0.48$, $2z/b = 0.92$ and $Re = 2 \times 10^5$ and $h/c = 0.315$ (a) Clean wing (b) Wing with CtSVGs.

An important concept in the design of the vortex generators is the dissipation distance of the produced vortices. The dissipation of the vortices should occur at the trailing edge of the main wing in order to avoid additional drag generation [Kuya et al., 2009]. Contours of vorticity in the x-direction at different downstream locations can be found in Figure 4.31. Flow separation

occurs at the trailing edge of the main wing. As mentioned, the vortices generated by the vortex generators should provide sufficient momentum for the boundary layer flow, and then dissipate at the trailing edge of the main wing in order to reduce additional drag generation. However, the vortices generated by the CtSVGs are still visible at $x/c = 1.052$, and the dissipation of the vortices occur at about $x/c = 1.315$. Reducing the size of the vortex generators may alleviate this issue.

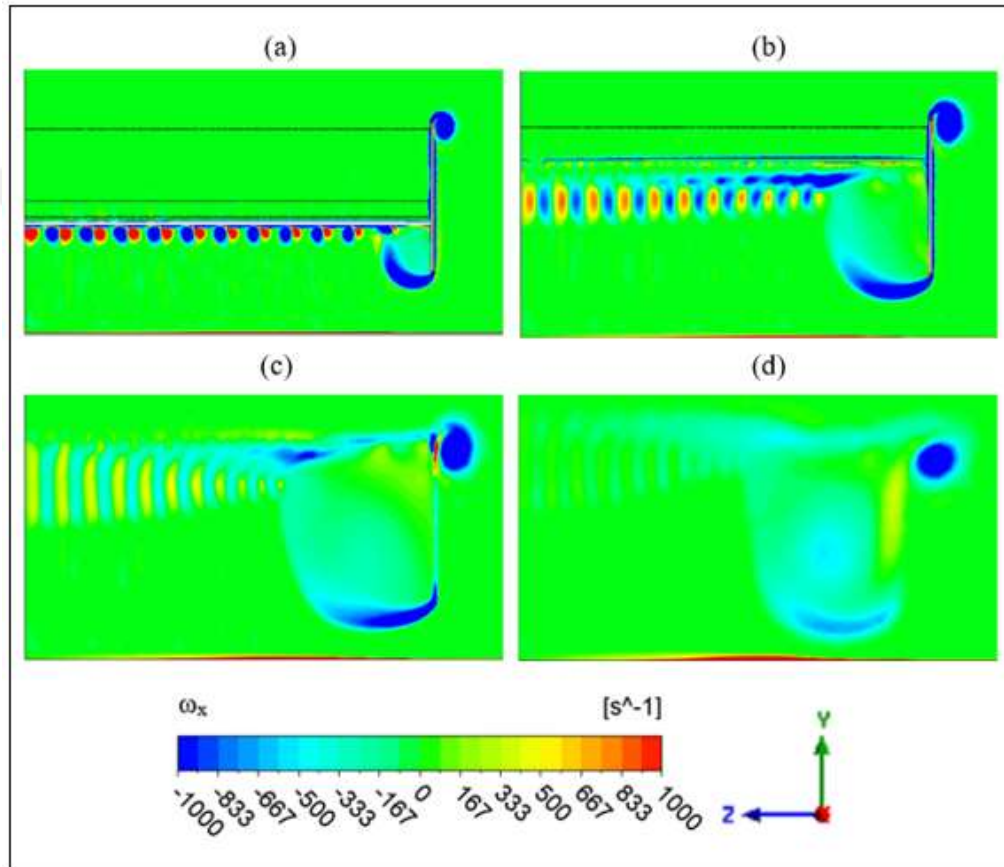


Figure 4.31: Contours of vorticity in x-direction viewed from downstream at $h/c = 0.211$ and $Re = 4.3 \times 10^5$ (a) $x/c = 0.526$ (b) $x/c = 0.789$ (c) $x/c = 1.052$ (d) $x/c = 1.315$.

These results provide insight into the effectiveness of the vortex generators under certain circumstances. The following conclusions for the phase 2 and phase 3 cases can be drawn:

- Vortex generators are able to eliminate or reduce the flow separations at $Re = 4.3 \times 10^5$ for all indicated ground clearances. Downforce generation is

increased by 25% - 30% with the addition of vortex generators at $Re = 4.3 \times 10^5$.

- Drag coefficient increases by 2% - 13% with the use of vortex generators at $Re = 4.3 \times 10^5$. The increase in drag may be reduced by using sub-boundary layer vortex generators. Avoiding the use of vortex generators at sections closer to the endplates may also be effective in reducing the drag generation.

- The CtSVG wing configuration cases at $Re = 2 \times 10^5$ exhibit unsteady behavior, and therefore the cases at $h/c = 0.211$ and $h/c = 0.263$ did not converge. However, vortex generators proved to be effective at $h/c = 0.315$ and $Re = 2 \times 10^5$. The flow separation is eliminated, and therefore, the downforce is increased by the addition of the vortex generators.

- Vortices generated by the vortex generators dissipate behind the flap of the wing at $Re = 4.3 \times 10^5$. Optimizing the vortex generator size may help with the dissipation distance, and therefore obtain less drag.

5. CONCLUSION AND FUTURE WORK

A three-dimensional CFD analysis of a multi-element front wing of a race car using various parameters such as ground clearance, Reynolds number and angle of attack is presented in this study. The cases are divided into three sections in order to analyze the aerodynamic behavior of the wing in a systematic way.

Phase 1 cases include simulations done at $AoA = 14.6^\circ$ on a clean wing configuration, and those cases do not exhibit flow separation. The downforce increases as the ground clearance decreases, and therefore, the flow reversal phenomenon is not observed in this section. The drag coefficient also increases as the ground clearance decreases. The dependency of the drag and downforce on the Reynolds number is negligible.

Phase 2 cases show the effects of increased AoA on downforce and drag. Those cases are done at $AoA = 22^\circ$ on a clean wing configuration. Flow separation can be observed in those cases. Downforce does not increase as the ground clearance decreases for those cases, and flow separation is prevented by operating at higher ground clearance and Reynolds number. Reynolds number has a significant impact on the downforce. At $h/c = 0.315$, the flow separation occurs at $Re = 2 \times 10^5$, whereas no separation is observed at $Re = 4.3 \times 10^5$.

Phase 3 cases show the effectiveness of vortex generators in eliminating or reducing flow separations. These cases are conducted at $AoA = 22^\circ$ on a CtSVG wing configuration. The flow separation is eliminated at $Re = 4.3 \times 10^5$ for all chosen ground clearances. At $Re = 2 \times 10^5$ and $h/c = 0.315$, a significant increase in downforce is observed with the addition of vortex generators. The addition of the vortex generators results in a higher drag coefficient compared to the clean wing configuration.

There are various methods which can be implemented in order to increase the accuracy of the results and the performance of the wing. An unsteady solver may be used to observe the unsteady behavior of the flow for the cases with vortex generators at $Re = 2 \times 10^5$. As for the performance of the wing, the dimensions of the vortex generators can be adjusted in order to minimize drag. Using vortex generators close to the endplate may also be avoided, since the flow separation occurs only around mid-span. This can result in a reduction of the wake size, and therefore less drag.

REFERENCES

- Agathangelou B., Gascoyne M., (1998), “Aerodynamic design considerations of a Formula 1 racing car”, SAE Technical Papers, 980399.
- Anderson J. D., (2007), “Fundamentals of aerodynamics”, 6th Edition, McGraw-Hill.
- Arrondeau B., Rana Z. A., (2020), “Computational aerodynamics analysis of non-symmetric multi-element wing in ground effect with humpback whale flipper tubercles”, *Fluids*, 5(4), 247.
- Bertin J. J., Cummings R. M., (2009), “Aerodynamics for Engineers”, 5th edition, Pearson.
- Çengel Y. A., Cimbala J. M., (2018), “Fluid Mechanics Fundamentals and Applications”, Fourth Edition, McGraw-Hill.
- Dominy R. G., (1992), “Aerodynamics of Grand Prix cars”, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 206(4), 267-274.
- Fackrell J. E., Harvey J. K., (1974), “The aerodynamics of an isolated road wheel”, PhD Thesis, Imperial College London.
- Flandro G. A., McMahon H. M., Roach R. L. (2011), “Basic aerodynamics: Incompressible flow”, 1st Edition, Cambridge University Press
- Galoul V., Barber, T. J., (2007), “A study of an inverted wing with endplates in ground effect”, Proceedings of the 16th Australasian Fluid Mechanics Conference, Gold Coast Australia, December.
- Garrood B. J., (2004), “Aerodynamic Interaction Mechanisms Relevant to the Underbodies of Formula 1 Cars”, PhD Thesis, Imperial College London.
- George A. R., (1981), “Aerodynamic effects of shape, camber, pitch, and ground proximity on idealized ground-vehicle bodies”, *Journal of Fluids Engineering, Transactions of the ASME*, 103(4), 631–637.
- Gunston B., (2005), “The cambridge aerospace dictionary”, *IEEE Aerospace and Electronic Systems Magazine*, 20(7), 35-36.
- Hain R., Kähler C. J., Radespiel R., (2009), “Dynamics of laminar separation bubbles at low-Reynolds-number aerofoils”, *Journal of Fluid Mechanics*, 630, 129-153.
- Hu X., Zhang R., Ye J., Yan X., Zhao Z., (2011), “Influence of different diffuser angle on Sedan’s aerodynamic characteristics”, *Physics Procedia*, 22, 239-245.

Jasinski W. J., Selig M. S., (1998), “Experimental study of open-wheel race-car front wings”, SAE Technical Papers, 983042.

Katz J., (1995), “Race Car Aerodynamics : Designing for Speed”, 2nd Edition, Bentley Publishers.

Katz J., (2006), “Aerodynamics of race cars”, Annual Review of Fluid Mechanics, 38, 27-63.

Kundu P. K., Cohen I.M., (2008), “Fluid Mechanics”, Fourth Edition, Academic Press.

Kuya Y., Takeda K., Zhang X., (2010), “Computational investigation of a race car wing with vortex generators in ground effect”, Journal of Fluids Engineering, Transactions of the ASME, 132(2).

Kuya Y., Takeda K., Zhang X., Beeton S., Pandaleon T., (2009), “Flow physics of a race car wing with vortex generators in ground effect”, Journal of Fluids Engineering, Transactions of the ASME, 131(12).

Launder B. E., Spalding D. B., (1974), “The numerical computation of turbulent flows”, Computer Methods in Applied Mechanics and Engineering, 3(2), 269-289.

Lin J. C., (2002), “Review of research on low-profile vortex generators to control boundary-layer separation”, Progress in Aerospace Sciences, 38(4–5), 389-420.

Mahon S., Zhang X., (2005), “Computational analysis of pressure and wake characteristics of an aerofoil in ground effect”, Journal of Fluids Engineering, Transactions of the ASME, 127(2), 290-298.

McLean D., (2012), “Understanding Aerodynamics: Arguing from the Real Physics”, 1st Edition, Wiley.

Menter F. R., (1994), “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA Journal, 32(8), 1598–1605.

Menter F. R., Langtry R. B., Likki S. R., Suzen Y. B., Huang P. G., Völker S., (2006), “A correlation-based transition model using local variables - Part I: Model formulation”, Journal of Turbomachinery, 128(3), 413-422.

Mitchell W. C., Staniforth A., Scott I., (2006), “Analysis of Ackermann steering geometry”, SAE Technical Papers, 2006-01-3638.

Mokhtar W. A., (2005), “A numerical study of high-lift single element airfoils with ground effect for racing cars”, SAE Technical Papers, 2005-01-0607.

Partridge J., (2021), “How do Formula 1 Front Wings Work?”, F1 Chronicle, April.

Pegrum J., (2006), "Experimental study of the vortex system generated by a Formula 1 front wing", PhD Thesis, Imperial College London.

Ranzenbach R., Barlow J., (1996), "Cambered airfoil in ground effect-an experimental and computational study", SAE Technical Papers, 960909.

Ranzenbach R., Barlow J. B., (1994), "Two-dimensional airfoil in ground effect, an experimental and computational study", SAE Technical Papers, 942509.

Ranzenbach R., Barlow J. B., (1995), "Cambered airfoil in ground effect - wind tunnel and road conditions", 13th Applied Aerodynamics Conference, San Diego USA, June.

Ranzenbach R., Barlow J., Diaz R., (1997), "Multi-element airfoil in ground effect - An experimental and computational study", 15th Applied Aerodynamics Conference, Atlanta USA, June.

Roberts L. S., (2013), "Boundary-Layer Transition on Wings in Ground Effect", PhD Thesis, Cranfield University.

Roberts L. S., Finnis M. V., Knowles K., (2016), "Characteristics of Boundary-Layer Transition and Reynolds-Number Sensitivity of Three-Dimensional Wings of Varying Complexity Operating in Ground Effect", Journal of Fluids Engineering, Transactions of the ASME, 138(9).

Schlichting H., Gersten K., (2000), "Boundary-Layer Theory", 8th Edition, Springer

Schubauer G. B., Skramstad H. K., (1947), "Laminar boundary-layer oscillations and transition on a flat plate", Journal of Research of the National Bureau of Standards, 38(2), 251.

Shih T. H., Liou W. W., Shabbir A., Yang Z., Zhu J., (1995), "A new $k-\epsilon$ eddy viscosity model for high Reynolds number turbulent flows", Computers and Fluids, 24(3), 227–238.

Smith C., (1978), "Tune to win the art and science of race car development and tuning", 1st Edition, Aero Publishers.

Soso M. D., Wilson P. A., (2006), "Aerodynamics of a wing in ground effect in generic racing car wake flows", Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 220(1), 1-13.

Soso M., Wilson P., (2004), "Investigating changes to the downforce curve of a double element airfoil in ground effect", SAE Technical Papers, 2004-01-3558.

Spalart P. R., Allmaras S. R., Reno J., (1992), "One-Equation Turbulence Model for Aerodynamic Flows", 30th Aerospace Sciences Meeting and Exhibit, USA, January.

Tallman J.W., (2017), “What am I? Vortex generators. Disrupting the air on purpose”, AOPA, August.

Tani I., (1964), “Low-speed flows involving bubble separations”, *Progress in Aerospace Sciences*, 5, 70–103.

Tennekes H., Lumley J. L., (2020), “A First Course in Turbulence”, MIT Press.

Versteeg H. K., Malalasekera W., Orsi G., Ferziger J. H., Date A. W., Anderson J. D., (1995), “An Introduction to Computational Fluid Dynamics - The Finite Volume Method”, First Edition, McGraw-Hill.

Walters D. K., Cokljat D., (2008), “A three-equation eddy-viscosity model for Reynolds-averaged Navier-Stokes simulations of transitional flow”, *Journal of Fluids Engineering, Transactions of the ASME*, 130(12).

White F. M., (2011), “Fluid Mechanics”, 7th edition, McGraw-Hill.

Wilcox D. C., (1986), “Multiscale model for turbulent flows”, *AIAA Journal*, 26(11), 1311–1320.

Wilson M. R., Dominy R. G., Straker A., (2008), “The Aerodynamic Characteristics of a Race Car Wing Operating in a Wake”, *SAE Technical Papers*, 2008-01-0658.

Yakhot V., Orszag S. A., (1986), “Renormalization group analysis of turbulence. I. Basic theory”, *Journal of Scientific Computing*, 1(1), 3–51.

Yang Z., Voke P. R., (2001), “Large-eddy simulation of boundary-layer separation and transition at a change of surface curvature”, *Journal of Fluid Mechanics*, 439, 305–333.

Zaki T. A., (2013), “From streaks to spots and on to turbulence: Exploring the dynamics of boundary layer transition”, *Flow, Turbulence and Combustion*, 91(3).

Zerihan J., Zhang X., (2000), “Aerodynamics of a single element wing in ground effect”, *Journal of Aircraft*, 37(6).

Zerihan J., (2001), “An Investigation into the Aerodynamics of Wings in Ground Effect”, PhD Thesis, University of Southampton.

Zhang X., Zerihan J. (2002), “Aerodynamics of a Single Element Wing in Ground Effect”, *Journal of Aircraft*, 37(6).

Zhang X., Zerihan J., (2003), “Aerodynamics of a Double-Element Wing in Ground Effect”, *AIAA Journal*, 41(6), 1007–1016.

Zhang X., Zerihan J., (2004), “Edge vortices of a double-element wing in ground effect”, *Journal of Aircraft*, 41(5).



BIOGRAPHY

Metin Baştuğ attended Cahit Elginkan Anatolian High School in Kocaeli. After graduating from the Mechanical Engineering Bachelor's degree program in Kocaeli University in 2015, he continued his studies in the Mechanical Engineering Master's degree program. Currently, he is working as a Mechanical Designer in an international company.

