



REPUBLIC OF TURKEY
ADANA ALPARSLAN TÜRKESİ SCIENCE AND TECHNOLOGY
UNIVERSITY

GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
DEPARTMENT OF AEROSPACE ENGINEERING

PASSIVE FLOW CONTROL AROUND THE AIRFOIL WITH SLOT

OMER FETHİ ASAN
MASTER OF SCIENCE



**REPUBLIC OF TURKEY
ADANA ALPARSLAN TÜRKESİ SCIENCE AND TECHNOLOGY
UNIVERSITY**

**GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
DEPARTMENT OF AEROSPACE ENGINEERING**

PASSIVE FLOW CONTROL AROUND THE AIRFOIL WITH SLOT

**OMER FETHİ ASAN
MASTER OF SCIENCE**

**SUPERVISOR
ASSOC. PROF. DR. TAHİR DURHASAN**

ADANA 2022



I hereby declare that all information in this thesis has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all information that is not original to this work.

[Signature]

Omer Fethi ASAN

ABSTRACT

PASSIVE FLOW CONTROL AROUND THE AIRFOIL WITH SLOT

Omer Fethi ASAN

Department of Aerospace Engineering

Supervisor: Assoc. Prof. Dr. Tahir DURHASAN

July 2022, 86 pages

In this thesis, numerical analysis was performed to control of flow over NACA 0018 airfoil with slot at variant angles of attack. The Reynolds number was set to $Re=3 \times 10^5$. Inlet of slot was posiniated on the pressure side of airfoil at chordwise location of $x/c=5\%$ while outlet of slot was located on the suction side of airfoil at three different chordwise location. Numerical analyzes were carried out using two different turbulence models, K- ω SST and K- ϵ RNG. Obtained results from the numerical analyzes were evaluated in terms of effect of the slot on flow structure and aerodynamic forces. It was observed that the location of slot remarkably affect flow over airfoil and delays stall angle. The model 40 suppress the formation of recirculation region formed over the airfoil. Furthermore, it was observed that while the stall angle of base airfoil is $\alpha=16^\circ$, it was delayed up to $\alpha=20^\circ$ for Model 40. Also, it was observed that the lift coefficient was increased by 154% for Model 40 at $\alpha=19^\circ$. On the other hand, it was observed that Model 20 significantly effective on the flow control in comparison with the base airfoil at angles of attack of $\alpha \geq 13^\circ$, while it has insignificant effect at low angles of attack. Similar observation was also obtained for Model 60 at low angles of attack. According to obtained results from both turbulence models, the most efficient model was determined as Model 40 in order to control flow around the airfoil especially for post stall angles.

Keywords: Airfoil, passive flow control, CFD, low Reynolds number, slot

ÖZET

KANAT ETRAFINDAKİ AKIŞIN YARIK İLE PASİF KONTROLÜ

Ömer Fethi AŞAN

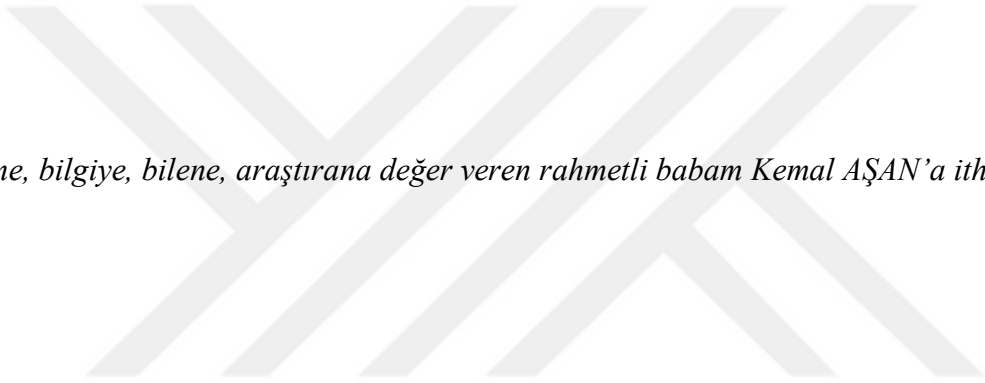
Havacılık ve Uzay Mühendisliği Anabilim Dalı

Danışman: Doç. Dr. Tahir DURHASAN

Temmuz 2022, 86 sayfa

Bu tez çalışmasında, NACA 0018 kanat profili etrafındaki akışın yarik ile kontrol edilmesinin değişken hücum açılarında sayısal analizi yapılmıştır. Reynolds sayısı $Re=3 \times 10^5$ olarak belirlenmiştir. Yarik girişi kanat profilinin basınç tarafında veter uzunluğuna göre $x/c=5\%$ lik kısmında konumlandırılırken, yarik çıkışı kanat profilinin emme tarafında üç farklı lokasyonda konumlandırılmıştır. Sayısal analizler, K- ω SST ve K- ϵ RNG olmak üzere iki farklı türbülans modeli kullanılarak gerçekleştirilmiştir. Sayısal analizlerden elde edilen sonuçlar, yarığın akış yapısı ve aerodinamik kuvvetler üzerindeki etkisi açısından değerlendirilmiştir. Yarığın konumunun, kanat profili üzerindeki akışı önemli ölçüde etkilediği gözlenmiştir. Model 40'ın, kanat profili üzerinde oluşan dolanımlı bölgesinin oluşumunu engellediği gözlemlenmiştir. Ayrıca yarıksız kanadın stall açısı $\alpha=16^\circ$ iken Model 40 modelinin bu açıyı $\alpha=20^\circ$ 'ye kadar ötelediği gözlemlenmiştir. Ayrıca Model 40'ın $\alpha=19^\circ$ hücum açısında kaldırma katsayısını %154 arttığı gözlemlenmiştir. Öte yandan Model 20'nin $\alpha \geq 13^\circ$ hücum açılarında yarıksız kanat profiline göre akış kontrolü üzerinde önemli ölçüde etkili olduğu, düşük hücum açılarında ise önemsiz bir etkiye sahip olduğu görülmüştür. Düşük hücum açılarında Model 60 için de benzer sonuçlar elde edilmiştir. Her iki türbülans modelinden elde edilen sonuçlara göre, özellikle stall açılarında kanat etrafındaki akışı kontrol etmek için en etkili modelin Model 40 olduğu gözlemlenmiştir.

Anahtar Kelimeler: Kanat, pasif akış kontrolü, düşük Reynolds sayılı akış, HAD, yarik



Eđitime, bilgiye, bilene, arařtırana deęer veren rahmetli babam Kemal AřAN'a ithafen...

ACKNOWLEDGEMENTS

First of all, I would like to thank my dear supervisor Assoc. Prof. Dr. Tahir DURHASAN, who helped me for my any question from the beginning to the end of this thesis study. Without the support of him, this thesis would not have been existed.

I sincerely thanks Assoc. Prof. Dr. Ilyas KARASU and Assist.Prof. Dr. Hurrem AKBIYIK who inspired me and gave advice to improve myself.

I would like to thank my friend, Emre GULER, who contributed to the preparation of this thesis.

I would like to special thank my dear friend Muhammet Yusuf ERGUN, who has touched almost every aspect of my life from my undergraduate education to the preparation of this Master's thesis, and has never spared his help and sacrifice.

I would like to thank Recai PELVAN, the boss of the workplace where I worked at the beginning of my graduate education, for the sacrifices he made at that time.

I would like to thank Onur INAN and Namık POYRAZ, who have induced to important turning points in my life.

I would like to thank Ahmed MENEVSEOGLU to extend my vision and for all his help.

I would like to thank my dear supervisor Prof. Dr. Taylan ALTAN, who gave me the chance to gain experience in the laboratory of the Department of Mechanical Engineering at Ohio State University.

I would like to thank my sister Secil ASAN for all her support. I would like to thank my beloved mother, Hacer ASAN who is behind every success I have. Without morale motivation and the vision her gave, this thesis would not have been written.

I would like to thank my dear wife Esra ASAN, who is always with me with self-sacrifice.

TABLE OF CONTENTS

TABLE OF CONTENTS	viii
LIST OF FIGURES	x
LIST OF TABLES	xii
NOMENCLATURE	xiii
1. INTRODUCTION	1
1.1. Flow Around An Airfoil	1
1.2. Laminar Separation Bubble	4
1.3. Flow Control Methods	6
1.4. Active Flow Control	8
1.5. Passive Flow Control	10
2. LITERATURE REVIEW	12
2.1. Active Flow Control	12
2.1.1. Synthetic Jets	12
2.1.2. Acoustic Excitation	13
2.1.3. Suction and Blowing	14
2.1.4. Oscillating	16
2.2. Passive Flow Control	17
2.2.1. Partially Flexible Wings	17
2.2.2. Cavity	17
2.2.3. Leading Edge Slats	18
2.2.4. Gurney Flap	19
2.2.5. Splitter Plate	21
2.2.6. Vortex Generators	22
2.2.7. Riblets and Roughness Surface	23
2.2.8. Flexible Membrane Wing	25
2.2.9. Leading Edge Serrations	27
2.2.10. Micro Small Cylinder	28
2.2.11. Slotted Airfoils	28

3. MATERIALS AND METHODS	33
3.1. Model and Computational Grid.....	34
3.2. Numerical Model and Boundary Conditions	39
3.3. Validation	40
4. RESULTS AND DISCUSSIONS.....	43
4.1. K-ω SST Model.....	44
4.2. K-ϵ RNG Model	57
4.3. Comparison of K-ω SST and K-ϵ RNG Turbulence Models.....	69
5. CONCLUSIONS	74
6. RECOMMENDATIONS.....	76
REFERENCES.....	77
CURRICULUM VITAE.....	86

LIST OF FIGURES

Figure 1.1. Lift Appearance by Velocity Differences.....	2
Figure 1.2. Differences between symmetrical and asymmetrical airfoil.....	2
Figure 1.3. Airfoil Shapes for Different Flow Type (Airplane Aerodynamics, 2021).....	3
Figure 1.4. Various Type of Airfoils (Wikipedia, 2021).....	3
Figure 1.5. Evolution of Boundary Layers.....	4
Figure 1.6. Explanation of Flow Separation.....	5
Figure 1.7. Laminar Separation Bubble (Katz, J., Plotkin, A., 1991, Low-Speed Aerodynamics from Wing Theory to Panel Methods, McGraw-Hill, Inc, 632 pp.)	6
Figure 1.8. Laminar Separation Bubbles on Different Section of an Airfoil	6
Figure 1.9. Flow Control Targets (Jahanmiri, 2010).....	7
Figure 1.10. Flow Control Methods.	8
Figure 1.11. Active Flow Control Triad.....	9
Figure 1.12. Some Passive Flow Control Methods	10
Figure 1.13. Delaying the Flow Separation by a Slot (Mohamed et al. 2019).....	11
Figure 3.1. The View of Flow Domain	35
Figure 3.2. Mesh Structure around base airfoil	35
Figure 3.3. The parameters of the Study	36
Figure 3.4. Mesh Generation of Slotted Airfoil	37
Figure 3.5: Sketching of Model 20	38
Figure 3.6: Sketching of Model 40	38
Figure 3.7: Sketching of Model 60	38
Figure 3.8: The Region that Separates $k-\omega$ from $k-\epsilon$	40
Figure 3.9: Mesh Independence Study.....	41
Figure 3.10. Comparison of Experimental Results and Present Results	42
Figure 4.1. Comparison of turbulence models with base airfoil at different angle of attack ..	43
Figure 4.2. Variation of lift Coefficient with the angles of Attack for different models	45
Figure 4.3. Variation of drag Coefficient with the angles of Attack for different models.....	46
Figure 4.4. Variation of lift to drag ratio with the angles of Attack for different models.....	47
Figure 4.5a. Pressure Coefficient distributions for all Models at $0^\circ \leq \alpha \leq 10^\circ$ ($k-\omega$ SST)	49
Figure 4.5b. Pressure Coefficient distributions for all Models at $12^\circ \leq \alpha \leq 19^\circ$ ($k-\omega$ SST)	50
Figure 4.6. Streamline Topologies for different Models at variant angles of attack.....	52
Figure 4.7. Stream-Wise Velocity Contours for Base, Model 20, Model 40 and Model 60 at $4^\circ \leq \alpha \leq 19^\circ$	54
Figure 4.8a. Comparison of Turbulence Kinetic Energy contours for at the Angles of Attack of $0^\circ \leq \alpha \leq 10^\circ$	55
Figure 4.8b. Comparison of Turbulence Kinetic Energy contour at the Angles of Attack of $12^\circ \leq \alpha \leq 19^\circ$	56
Figure 4.9. Variation of lift Coefficient with the angles of Attack for different models	58

Figure 4.10. Variation of drag Coefficient with the angles of Attack for different models....	59
Figure 4.11. Variation of lift Coefficient over drag Coefficient with the angles of Attack for different models.....	60
Figure 4.12a. Pressure Coefficient distributions for all Models at $0^\circ \leq \alpha \leq 10^\circ$ (k- ϵ RNG).....	61
Figure 4.12b. Pressure Coefficient distributions for all Models at $12^\circ \leq \alpha \leq 19^\circ$ (k- ϵ RNG) ..	62
Figure 4.13. Streamline Topologies for different Models at variant angles of attack.....	64
Figure 4.14. Stream-Wise Velocity Contours for Base, Model 20, Model 40 and Model 60 at $4^\circ \leq \alpha \leq 19^\circ$	66
Figure 4.15a. Comparison of Turbulence Kinetic Energy at the Angles of Attack of $0^\circ \leq \alpha \leq 10^\circ$	67
Figure 4.15b. Comparison of Turbulence Kinetic Energy at the Angles of Attack of $12^\circ \leq \alpha \leq 19^\circ$	68
Figure 4.16. Comparison of Coefficient of Pressure (CP) for Base and Model 40.....	70
Figure 4.17. Streamlines topologies obtained from K- ω SST and K- ϵ RNG turbulence models for $13^\circ \leq \alpha \leq 16^\circ$	71
Figure 4.18. Comparison of Turbulence Kinetic Energy (TKE).....	72

LIST OF TABLES

Table 3.1. Mesh Quality Table	34
Table 3.2. Test Cases of Slotted Airfoil	37
Table 3.3. Validation of Slotted Airfoil by comparison to the literature.....	42



NOMENCLATURE

Re : Reynolds number
 α : Angle of Attack
 δ : Boundary Layer Thickness
 C_L : Lift Coefficient
 C_D : Drag Coefficient
 C_L/C_D : Lift-to-Drag Coefficient
 C_p : Pressure Coefficient
 c : Chord Length
 P : Pressure
 ρ : Density
 V_∞ : Free-Stream Velocity
 ω : Specific Dissipation Rate
 $L1$: Distance To Slot Outlet From Leading Edge
 $L2$: Distance To Slot Inlet From Leading Edge
 μ : Molecular Viscosity
 θ : Momentum Thickness
 Tu : Turbulence Intensity
 y^+ : Dimensionless Wall Length
 $W1$: Outlet of the Slot Width
 $W2$: Inlet of the Slot
 R : Slot Edges
 ε : Dissipation of Turbulent Kinetic Energy

1. INTRODUCTION

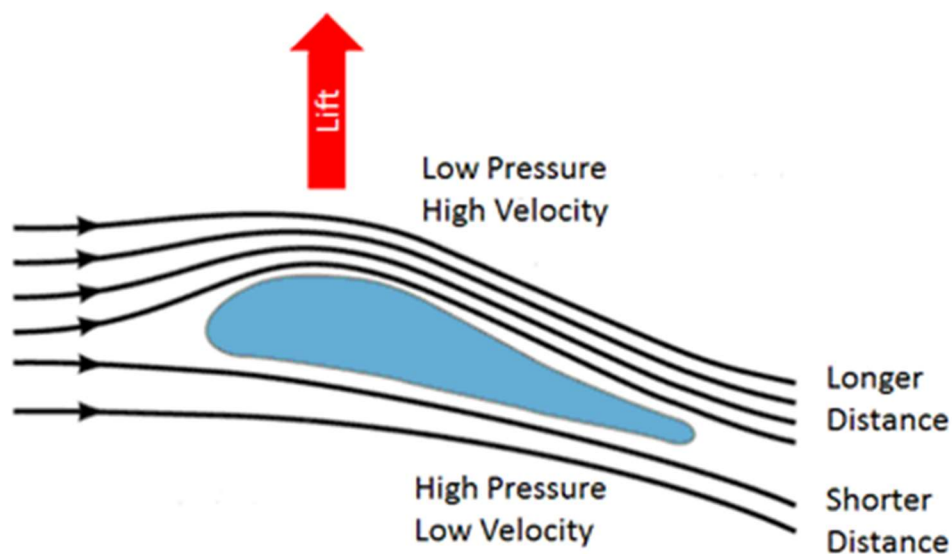
Different types of flow regimes appear around an airfoil and they can be categorized as laminar, transition and turbulent flows. If the Reynolds number is smaller than 500,000, the flow is accepted as low Reynolds number flow. The main characteristic of low Reynolds number flow is formation of laminar separation bubble, which occurs between separation and reattachment points because of the viscous effects or strong adverse pressure gradients. When the Reynolds number is low, the maximum lift force decreases. Nevertheless, there are a lot of insects, birds, etc. can fly easily in the air at low Reynolds number. Unmanned aerial vehicles (UAV), micro air vehicles (MAV), wind turbine blades, model planes, etc. are some of the examples in the engineering applications for low Reynolds number flow. Also, studying on low Reynolds number is getting important, with respect to the Mars mission due to the fact that Mars' atmosphere has lower density and temperature than the Earth's atmosphere.

In these application areas, different flow regimes can be occurred. For this reason, well understanding of the flow regimes are important issue in order to avoid undesired effects on the engineering design. For example, formation of the separation bubble over an airfoil and the scope of preventing it is the topic that many researchers keen on. Furthermore, the effect of transition and turbulent flow on the unmanned aerial vehicles' wing, wind turbine blades gets attention by many researcher.

1.1. Flow Around An Airfoil

The duty of an airfoil is generating lift by pressure difference between upper and lower surface of airfoil. When flow occurs around an airfoil, flow velocity gets increased in the upward side, while decreased in the downward side by the design of it. Hence, a velocity differences occur due to the fact that they arrive in the exit point at the same time, although fluid travels different distance on the upper and lower side of airfoil. Consequently, the pressure gradient is occurred according to Bernoulli principle. Bernoulli principle states that there is an inverse proportion between velocity and pressure. In other words, where fluid velocity increases, pressure decreases. The Figure 1.1 demonstrates that the how occurrence of lift force when flow occurs around an airfoil.

The design of an airfoil varies depending on the amount of force the wing needs to lift. As it is shown in Figure 1.2, they can be classified according to their geometry, which are symmetrical and asymmetrical. Symmetrical airfoils have the same shape between the up and down side of itself, while asymmetrical airfoils are consist of two different shapes. Moreover, up side of an airfoil is called as suction surface, while down side is called as pressure surface.



Also known as the "Longer Path" or "Equal Transit" Theory

Figure 1.1. Lift Appearance by Velocity Differences (www.mpoweruk.com/flight_theory)

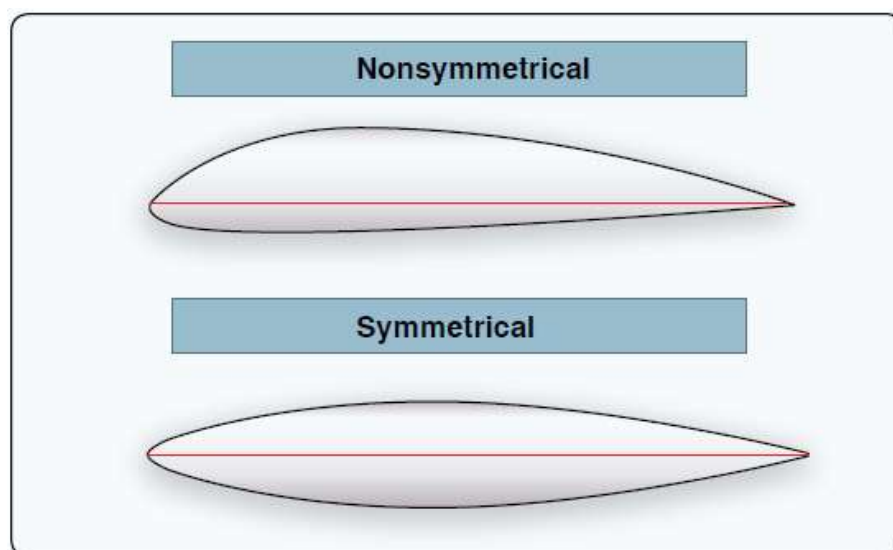


Figure 1.2. Differences between symmetrical and asymmetrical airfoil (www.flight-study.com/2019/12/airfoil-terminology-definitions)

Optimization is one of the important area that engineers study in every designed products. It is of great importance in wing design as it is in every area of design. Depending on the application area that an airfoil is going to be used, there are several designs of it. In the determination of eligible airfoil for the application area, some preferences are made. For example; the priority is to be higher lift force for sailboats while it is to be lower drag force for wings of supersonic jet engines.

Figure 1.3 and 1.4 present various types of airfoils encountered in the nature and industry application. Each of them has different geometries, such as chord length, leading edge radius and maximum camber.

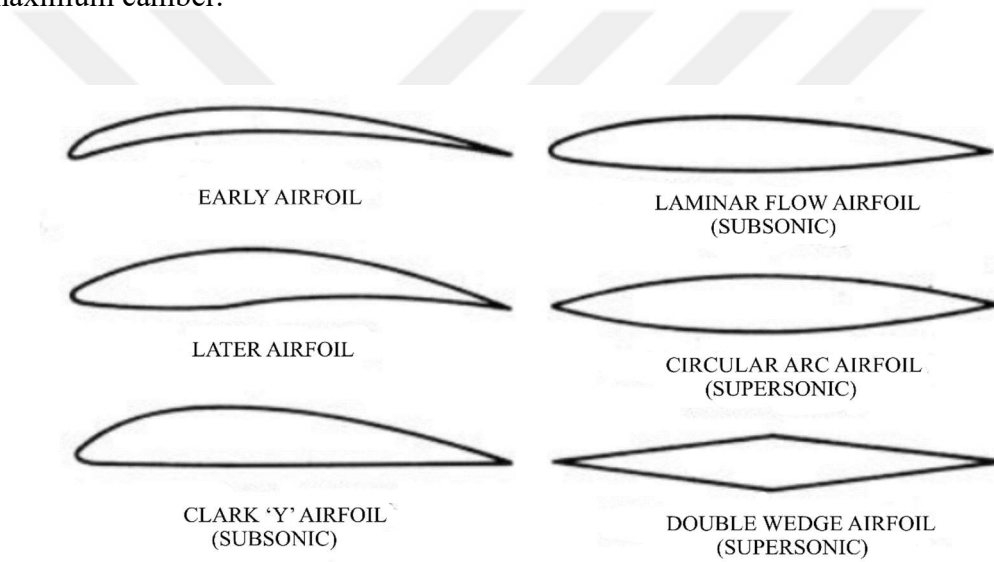


Figure 1.3. Airfoil Shapes for Different Flow Type (Airplane Aerodynamics, 2021)

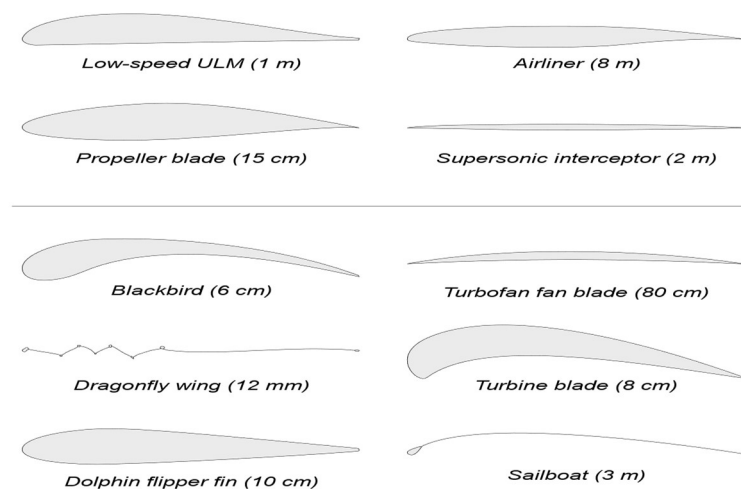


Figure 1.4. Various Type of Airfoils (en.wikipedia.org/wiki/Airfoil)

In Figure 1.5 the evolution of boundary layers (laminar and turbulent) are showed. The boundary layer is a thin surface just near to surface, where inertial forces are not dominant to viscous forces. The boundary layer can be categorized by the local Reynolds number such as laminar boundary layer, transition boundary layer and the turbulent boundary layer. The laminar boundary layer is stable. The flow gradually becomes unstable with increase in the Reynolds number. In other words, the transition region from laminar to turbulent forms. For relatively higher Reynolds number turbulent boundary layer occurs. In this region the stability of flow deteriorates completely and flow becomes unsteady, moves randomly.

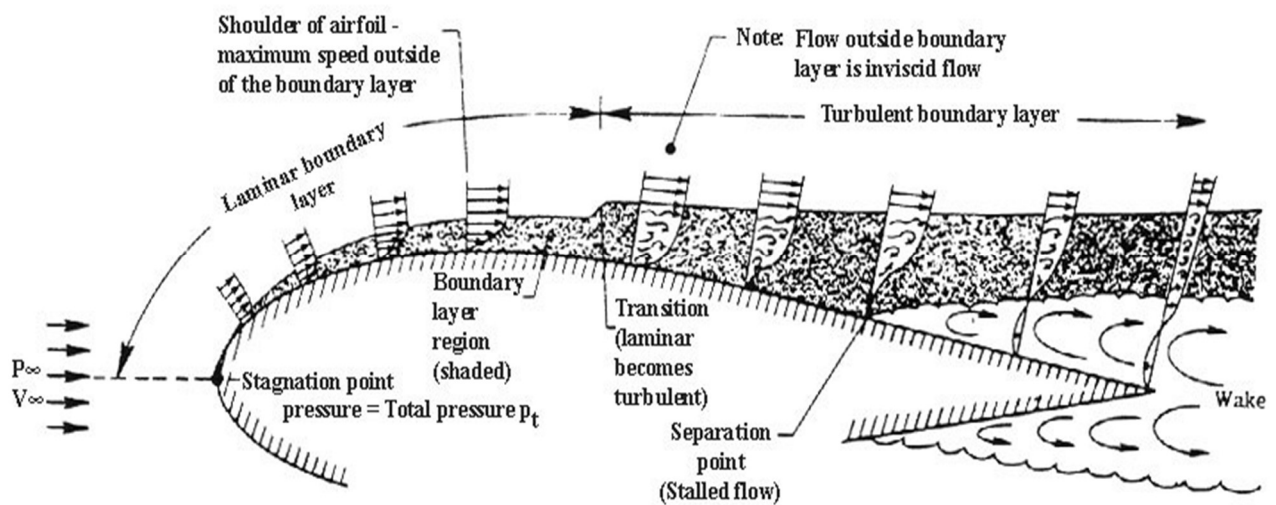


Figure 1.5. Evolution of Boundary Layers (U.S. Centennial, 2021)

1.2. Laminar Separation Bubble

In a specified region, because of the domination of viscous effects, the adverse pressure gradient forms and the flow separates from surface of the body. As soon as the separation happens, laminar flow becomes to turbulent flow and this region is called as transition region. As it is demonstrated in figure 1.6 there are three different parts to explain flow separation. At first flow accelerates, the pressure drops at this time. Subsequently, flow begins to lose its momentum while the adverse pressure increases. At the final point of this part, flow starts to separate. At the final part, after the flow separation, pressure stays constant. The flow that increasing energy with the transition to turbulent flow.

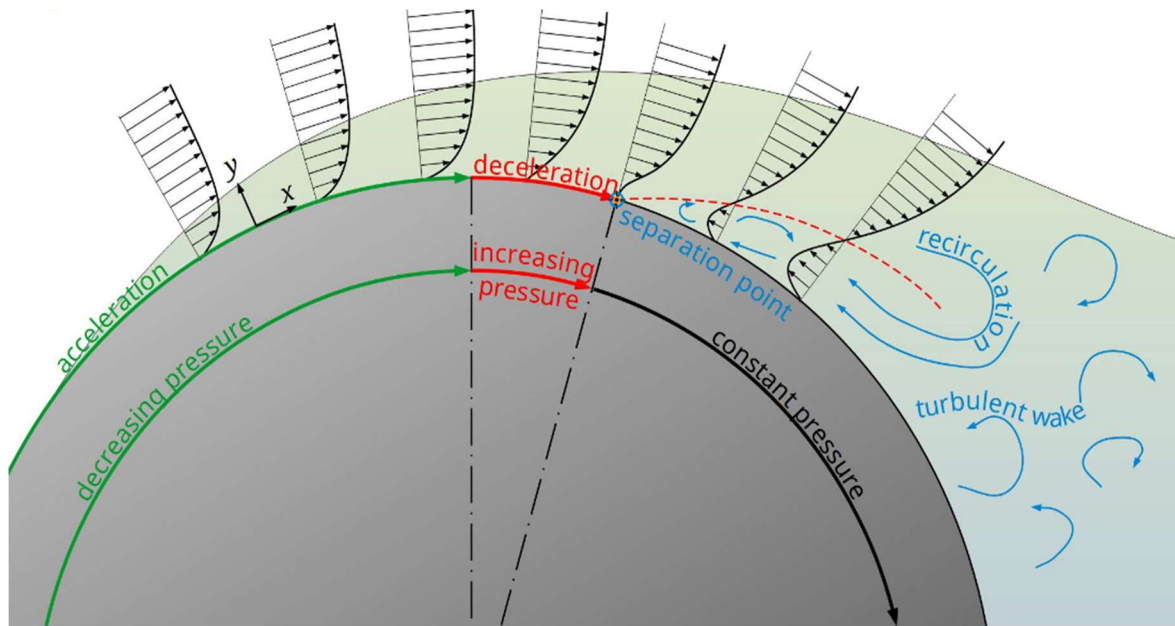


Figure 1.6. Explanation of Flow Separation (www.tec-science.com/mechanics/gases-and-liquids/flow-separation-boundary-layer-separation)

Figure 1.7 presents formation of laminar boundary layer separation and separation bubble. Laminar boundary layer separates from the surface and then transition from laminar to turbulent occurs so energy of flow increases. Turbulent flow attaches again to the surface of the body. A reverse flow region occurs in the bubble. The pressure coefficient distribution over the airfoil indicates that a hump appears. The beginning and end of the hump signifies separation and reattachment of the flow as depicted in figure 1.7. Laminar Separation Bubble (LSB) can be classified into two parts, short and long bubbles.

The position and size of the bubble are a function of angle of attack, flow disturbances, airfoil geometry, and Reynolds number. (Swift, 2009; Tani, 1964). As the angle of attack increases, the small size LSB moves and contracts in the direction of flow (Tani, 1964). A small region of constant pressure can be seen in the pressure distribution curve inside such a bubble. The movement of the split point towards the leading edge continues in direct proportion with the increase in the angle of attack. At a given angle of attack, the flow can no longer reattach to the surface. This situation is called breakdown or burst of the bubble. The appearance of deterioration does not lead to complete separation of the flow. The separated flow passes over the airfoil and reconnects to the flow below. The flow region below this separated flow is called the long bubble. (Figure 1.8)

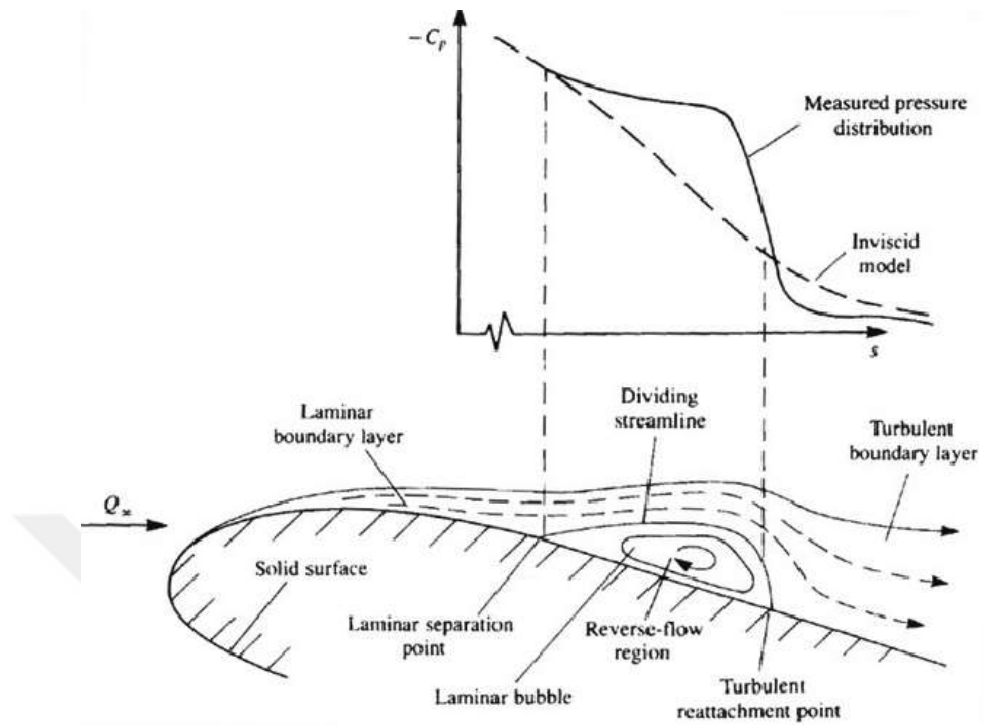


Figure 1.7. Laminar Separation Bubble (Katz J., Plotkin A., 1991.)



Figure 1.8. Laminar Separation Bubbles on Different Section of an Airfoil (Auld D.J., Srinivas K., 2008)

1.3. Flow Control Methods

The flow that occurs around an airfoil, separates, when it does not attach to the surface due to decrease in the momentum, increasing adverse pressure gradient, etc. as is mentioned above. With increasing the angle of attack, separation location on the suction surface moves to the leading edge of airfoil. Furthermore, stall condition occurs at higher angle of attacks and results in decreasing of lift force and increasing of drag force. Figure 1.9 shows the effects that occurrence of flow separation on an airfoil can cause. On the other hand, if an airfoil, which is exposed to having adequately Reynolds number flow, the flow separates lead to formation of vortex shedding. This causes a significant pressure drop on the rear surface of the profile. In addition, vortex shedding causes severe structural vibrations, resonance, and increases in average lift and drag fluctuations. While it can be counted among the advantages of facilitating

heat transfer by increasing turbulence, its main disadvantages are that it causes excessive vibration and ocustic noise.

Therefore, the study on flow control is keen on many researchers, in order to prevent or reduce undesirable effect of flow. In this content, many control methods have been applied to control of vortex shedding that occurs around airfoils. Some examples; base bleed, splitter bleed, suction and blowing electromagnetic control, etc.

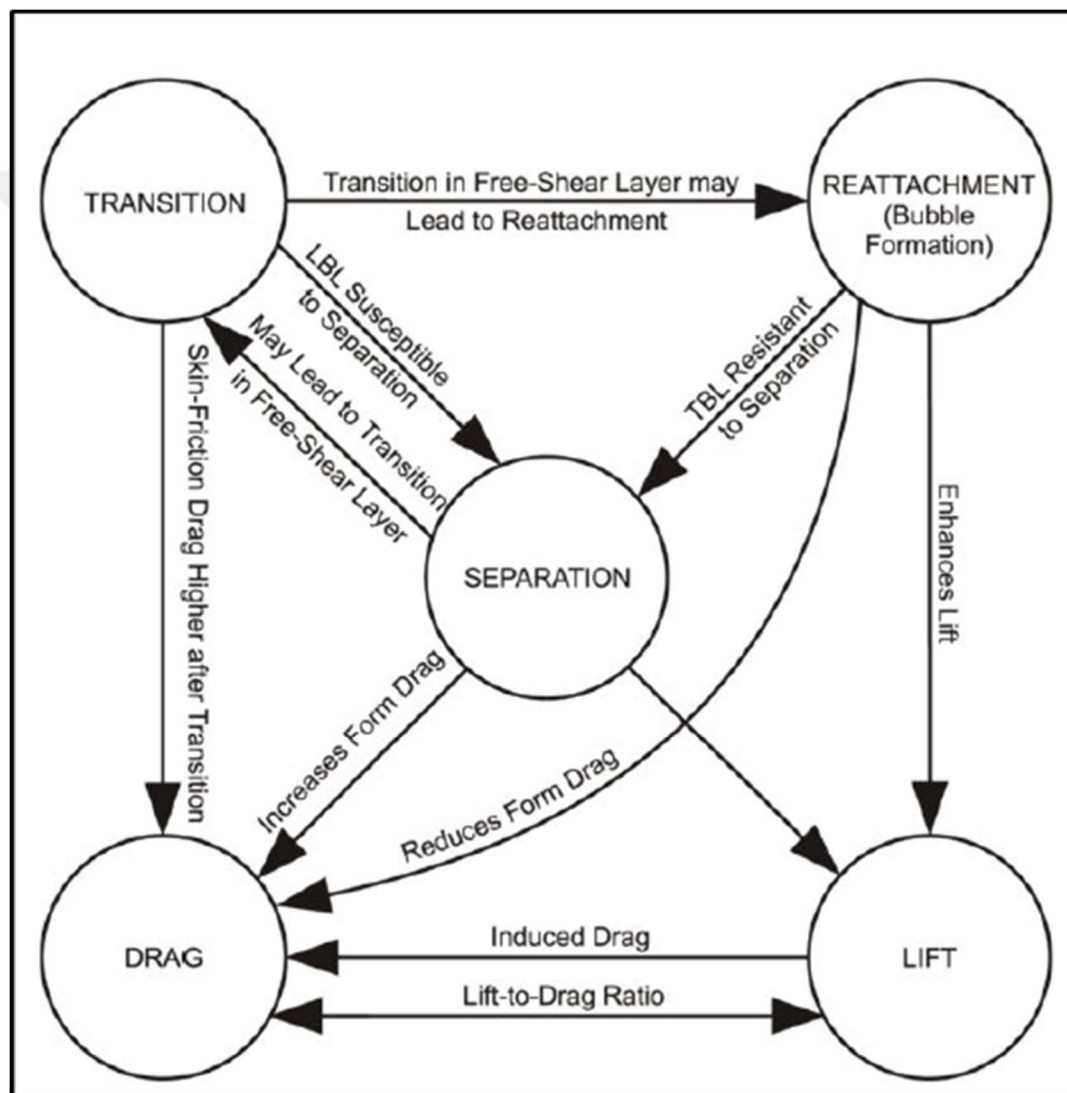


Figure 1.9. Flow Control Targets (Jahanmiri, 2010)

As is demonstrated in Figure 1.10, the flow control methods are divided into two group called as active and passive flow control methods. Passive control methods do not need to get external energy. Some examples for passive control method can be given as vortex generators, roughness surface, rod, splitter plate, etc. The passive control techniques based on the principle

changing geometry or adding control device into flow field in order to alternate boundary layer or wake region characteristics.

On the other hand, the external energy is needed in active flow control techniques. Synthetic jets, piezoelectric wings, microjets and plasma actuators is some of the examples for active control techniques. Active flow control is applied in many engineering applications such as gas turbines and propulsion systems.

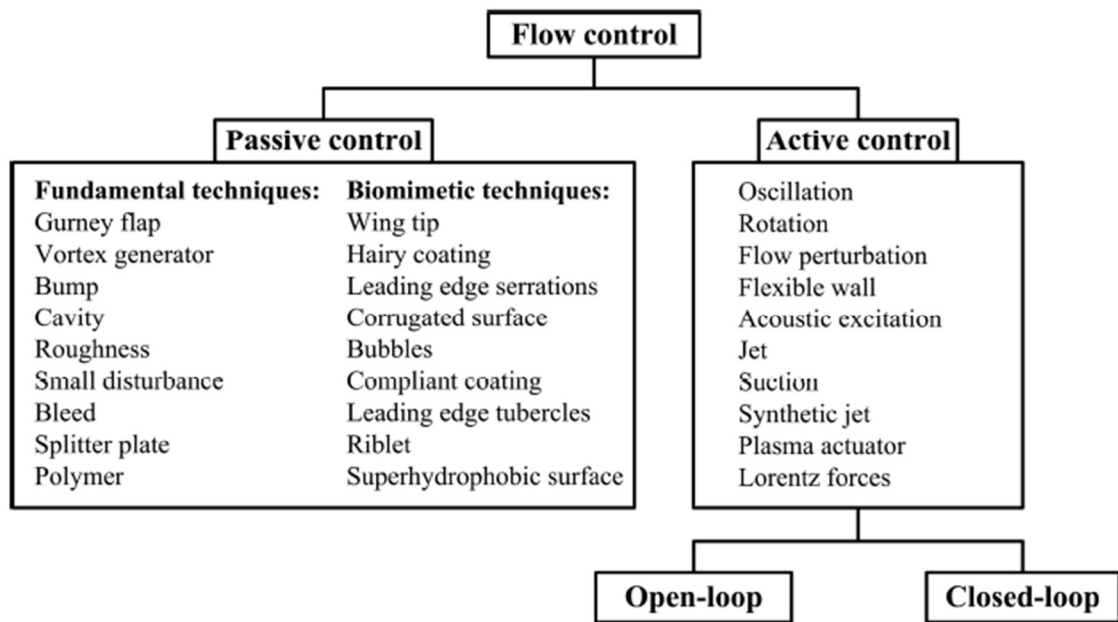


Figure 1.10. Flow Control Methods (Moghaddam and Neishabouri, 2017)

1.4. Active Flow Control:

The active control systems are also categorized as open and closed loop. In general, frequency and amplitude is set manually in the open loop active control systems and there is no feedback scheme on this control type. The common feature of closed loop active control system is that the change of the control signal through the feedback scheme by directly sensing a certain amount of fluid. By creating a change in the flow dynamics in the area where they are applied, improving their performance and saving energy are their most important advantages. The system with closed loop flow control, categorized as semi-static and dynamic feedback systems within their own. The semi-static closed loop control is used in systems with non-linear fluid dynamics. In other words, it is used in the time that the feedback time scale is bigger than the response time scale of the system. In dynamic closed-loop control, this is both time scale can be considered equal to each other.

The goal of active flow control is increasing lift and/or reducing drag and suppressing the vortex shedding. There are many kind of active flow control techniques applied in previous studies. Some of those; suction and blowing [Junxuan and Xun (2015); Kamari et al (2018)], actuators [Brunton (2014)] and jet flows [Zha (2007); Amitay (2001)], etc. Some example for active control methods applied different flow regimes are presented in Figure 1.11. The main benefit of the active control method is that it can perform the control at the desired place at any time. In contrast, the using of energy, in active control method is taken into account one of the deficits of the control method.

Studies on active flow control methods point out that a single flow control method cannot provide the desired performance in all flow applications. The zero net mass flow actuators are suitable for closed loop due to their wide operating frequency range and fast response capability, but may not meet the user's expectations with control dominance and frequency limits that may be insufficient in some high speed applications. On the other hand, resonance tubes are also preferred in high frequency applications, but they do not have the feature of keeping the frequency range and response time required for feedback under control. In this context, studies on flow control should be continued and control capabilities, limitations and costs should be determined exactly.

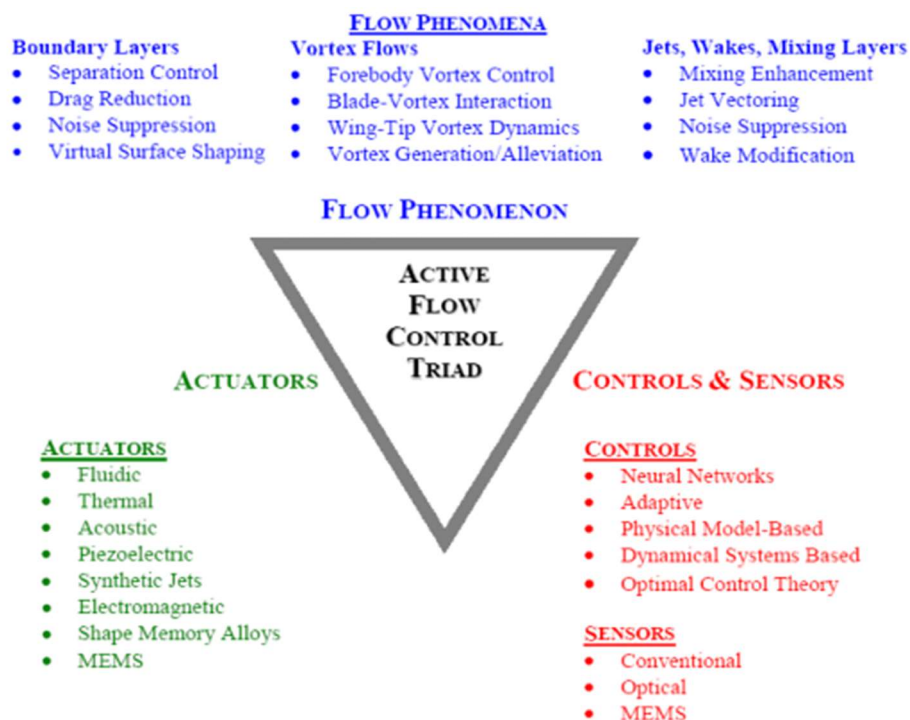


Figure 1.11. Active Flow Control Triad (Kral, 2000)

1.5. Passive Flow Control:

Passive flow control systems does not need energy source. The system can be manufactured easier and cheaper than active flow control systems. The biggest disadvantage of passive flow control systems is that they cannot be adapted to different conditions and do not always give good results. Same example for passive flow control methods can be given as cavities, bumps, slots, a cylinder that is placed in the boundary layer, spoilers, etc. are. It is possible to change the flow from laminar to turbulent by using rough surfaces or a flow stagger. In turbulent regime, the flow is more resistant to separation due to higher momentum. The separation of the flow from the surface can be postponed by increasing the surface roughness of the body or by using flow deflectors. Thus, the trace area formed behind the object is narrowed and it is possible to reduce the pressure. (Cengel and Cimbala, 2007). In addition, the drag forces can be reduced by reorganizing the geometry of the body, rounding the sharp corners or opening channels where necessary.

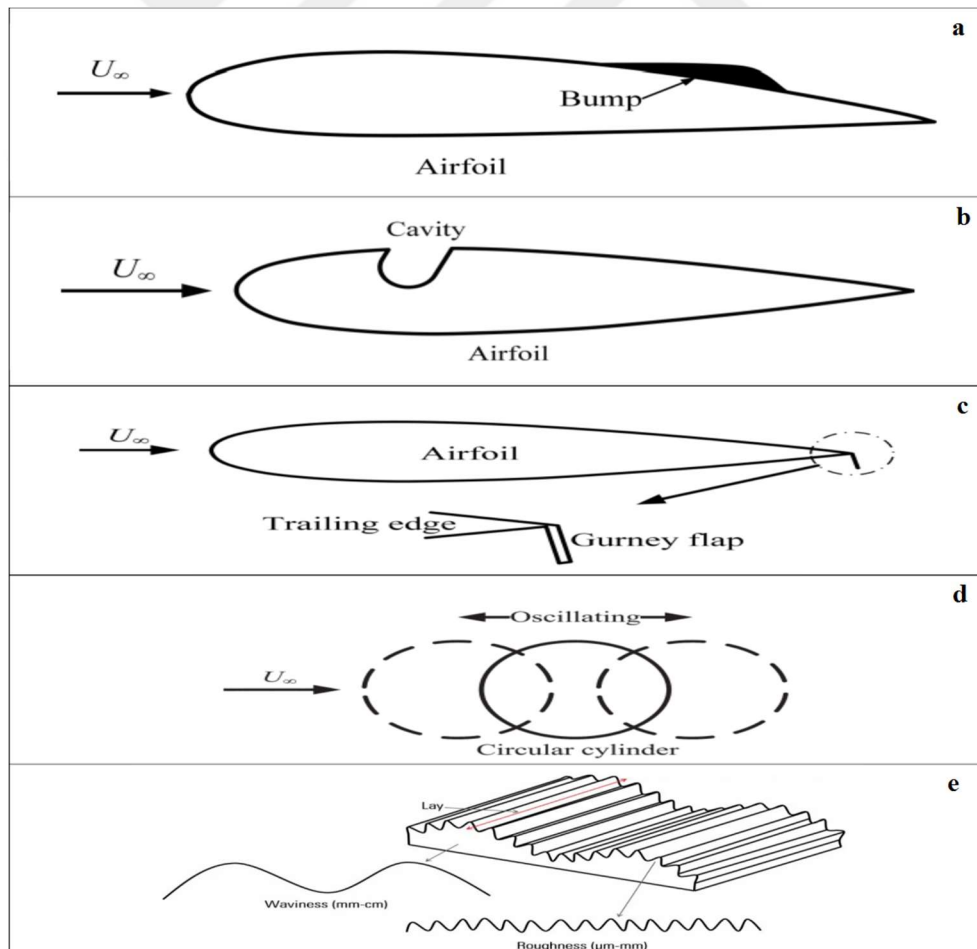


Figure 1.12. Some Passive Flow Control Methods (a,b,c: Wang and Feng, 2019; d,e: Cambridge University Press Flow Control Techniques and Applications, 2018)

There are a lot of passive control methods. For instance; roughness surface [Zhang et al. (2018); Wu et al. (2019)], gurney flap [Storms and Jang (1994); Yee et al. (2007)], vortex generators [Kerho et al. (1993); Lin et al. (1994)], bump and cavity [Fatehi et al. (2019); Raj Mohamed et al. (2019)] etc. and also these studies are in use on plenty of designs. Another passive control method is creating a slot. Mohamed et al. (2019), is showed that a slot delays the flow separation occurred over an airfoil. The contribution of a slot on delaying of the flow separation is shown in Figure 1.13.

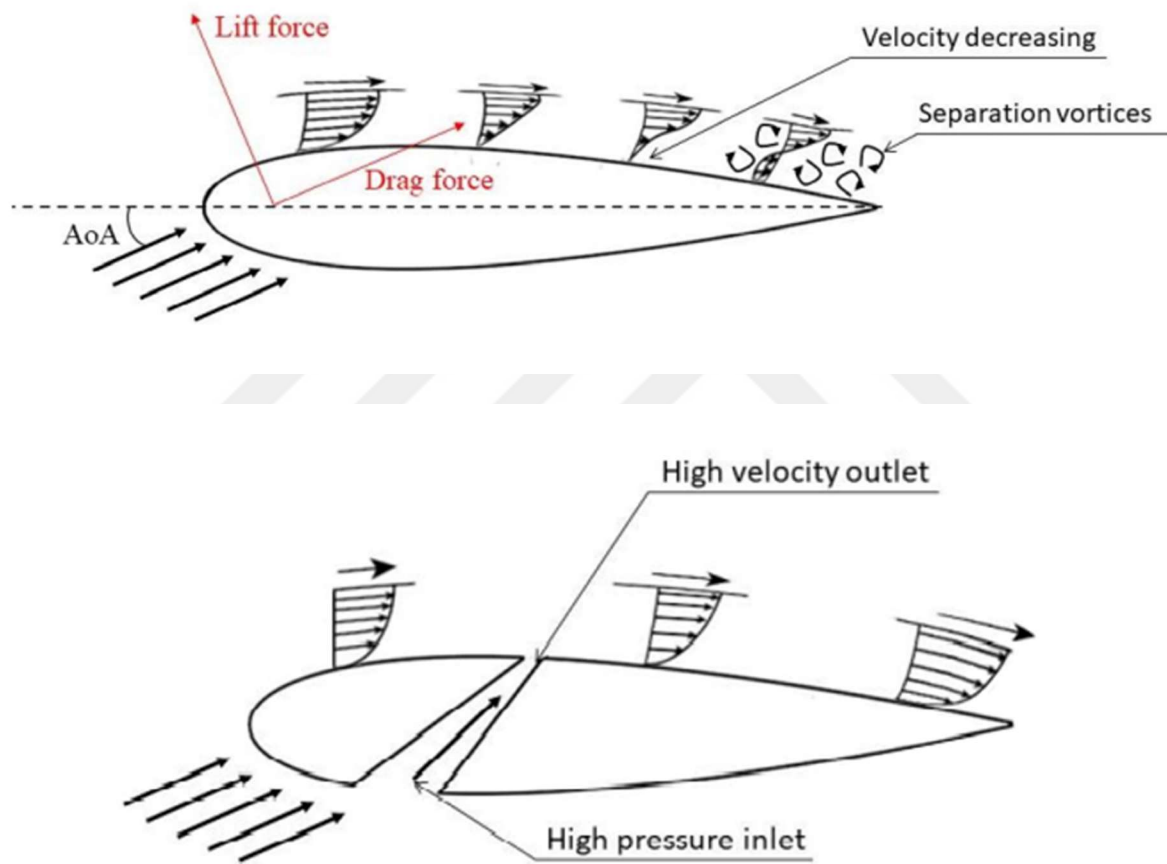


Figure 1.13. Delaying the Flow Separation by a Slot (Mohamed et al. 2019)

In this study, a NACA 0018 airfoil, that is slotted from different chordwise location, was analyzed numerically at the Reynolds number of $Re=3 \times 10^5$ at the angles of attack of $0^\circ \leq \alpha \leq 20^\circ$. The provided results were compared with base airfoil due to reveal the effect of slot on flow characteristic and aerodynamic performance. Three different models were created and will be discussed to provide the optimum performance. It is also aimed that increasing the lift coefficient and delaying the stall angle by using slotted airfoil.

2. LITERATURE REVIEW

2.1. Active Flow Control

2.1.1. Synthetic Jets

Jet and Vortex Actuator (JaVA) is a mechanically driven net zero mass flow, fluidic based active control mechanism. Synthetic jets are also called “net zero mass flow actuators” and appear in many flow applications. The general working principle is to provide periodic vibration by means of a piston or diaphragm and to pump the air in and out of the cavity containing a slit or orifice. As a result, vortex pairs are formed. These vortex pairs coalesce into a jet with a total of zero net mass flow transfer, so momentum is derived from the surrounding air.

Tousi et al. (2021) employed synthetic jet on Selig-Donovan 7003 (SD7003) airfoil at Reynolds number $Re = 6 \times 10^4$ due to qualify the lift and drag forces. Five different parameters were noted: jet position, jet width, momentum coefficient, forcing frequency and jet inclination angle. Moreover, four different angle of attacks (AoA) were considered: 4° , 6° , 8° and 14° . The aerodynamic efficiency was raised by 251% from base airfoil by carrying out a medium momentum coefficient at 14° angle of attack. However, only 39% increase was provided at 8° angle of attack.

Zong et al. (2018) performed plasma synthetic jet actuators (PSJA) on NACA 0015 airfoil model for controlling leading edge separation (LES) at Reynolds number $Re = 1.7 \times 10^5$. The lift coefficient was raised by 21%. Moreover, the stall angle of the airfoil was delayed by around 6.5° . With actuation frequency of $F^* \leq 0.25$, the separation point at $\alpha = 15.5^\circ$ stayed near the leading edge and the zero-velocity line was periodically swept to the suction region by spreading spanwise vortices that was generated from PSJA. For separation control, at $\alpha = 22^\circ$, the separation point was repeatedly existing upstream in the position of actuation, inattentive of frequency of the actuation.

Maldonado et al. (2021) used 60 synthetic jet actuators on NACA 0012 at nearly Reynolds number of $Re = 1 \times 10^6$. Three - 2.58 meters in diameter bladed rotor was tested at the angles of 2° , 5° and 8° , at speeds of 250, 500, 750 and 1000 RPM (revolutions per minute). They used high capacity load cells and current sensors due to evaluate the rotor thrust and power. They

also used Laser Doppler Velocimetry (LDV) techniques throughout the suction region of the airfoil. Two different flow control patterns were examined and checked: constant and non-constant momentum coefficients. At 250 RPM, when all 20 synthetic jets are activated for a blade, the constant momentum coefficient, was observed that it develops in low speed rotor performance. The non-constant momentum coefficient, moreover, was monitored that it improves at higher rotor speeds. They also noted that when both flow control patterns are carried out for a blade module at the rotor speeds of 750 and 1000 RPM, performance of the airfoil is better than baseline.

Lindstrom et al. (2018) experimentally investigated the interaction that between the 9 synthetic flow jets and the flow around a NACA 0012 airfoil. The cross-section of the test tunnel was 0.8 m x 0.8 m x 5 m. Reynolds number was set to $Re = 72 \times 10^5$. For discovering the three-dimensional flow structures, which were created because of the interaction, particle image velocimetry (PIV) measurements were provided. After jets were powered, the separated flow around the controlled area has become reattached and the lift coefficient was raised by 65%.

2.1.2. Acoustic Excitation

One of the active flow control method is acoustic excitation. The working principle of the method is sending a sound wave via a loudspeaker, etc. to the flow control area. Subsequently, the spended sound waves induce the flow. The induced field changes the boundary layer conditions and the flow is controlled.

Probsting et al. (2021) performed some experiments on NACA 0018 at Reynolds number $Re = 1.15 \times 10^5$ and they set the angle of attack to 5° . The receptivity of the flow and reply to tonal sound were tested by applying controlled transient and steady acoustic excitation. The controlled acoustic disturbances were supplied by a loudspeaker. Magnification of shear layer disturbances in the front region of the separation bubble was monitored just after some time lag from the beginning of acoustic force. In the reception area, the acoustically excited disturbances reached higher amplitudes than native perturbations. They were also greatly enlarged in the separation bubble, resulting in a transient reduction in the size of the separation bubble. Provided results indicated that the acoustic excitation is a notable effective way to control flow structure.

Andan et al. (2018) experimentally examined the effects of acoustic excitations on the noise spectrum. For the experiment, NACA 0015 was used and Reynolds number was set to $Re = 10^4$ due to measure the noise spectrum at low Reynolds number. Also, the angle of attack (AoA) was adjusted to three different angles; 0° , 3° and 5° . Several separate tones on excitation frequency of 2000 Hz and 3000 Hz were examined. In addition to this, the affect of excitation amplitude of 70 and 90 dB were investigated. It is found that the primary frequency is highly dependent on freestream velocity and the angle of attack. There was no change on secondary frequency. In summary, the investigation indicated that the only affected frequency was the primary one by acoustic excitation.

Suzuki and Ishii (2000) employed acoustic excitation on the discreted flows at Reynolds number $Re = 10^5$ and the effect of this control method was examined in two dimensional numerically simulating. Several angle of attack has been tested on NACA 0012 2D airfoil for this simulation; 10° , 12° , 14° , 16° . The results of the simulations indicated that the acoustic wave with convenient frequency vigorously depress the leading edge division. This case was found related to the vorticity structure, where the laminar separation point.

Yarusevych et al. (2007) experimentally examined the effect of acoustic excitation amplitude on boundary layer for NACA 0025 at low Reynolds number $Re = 1 \times 10^5$ and 1.5×10^5 . The experiment was applied at the 5° angle of attack (AoA). The pressure level of the noise was investigated for 105 dB and 106 dB and the excitation frequencies were 165 Hz and 455 Hz. Provided consequence indicated that the acoustic excitation is notably effective on the flow structure and there are some decreases of boundary layer thickness and wake area because of excitation.

2.1.3. Suction and Blowing

Suction and blowing active control method is applied for compressing or preventing unwanted flow structure around airfoils via sucking or blowing the flow. The blowing in trailing edge has important effect on reducing the drag force in general. The method is carried out to compress the separation via increasing the momentum of the boundary layer. This method helps the flow to have energy. Just after the flow energized, it encumbers boundary layer to be thickened and separation happened. All these above causes to reduced drag.

Abbasi and Souri (2020) aimed to investigate a two-dimensional flow around a rod-airfoil by using Unsteady Reynolds-Averaged Navier–Stokes (URANS) equations. They employed the suction and blowing active control method due to control the flow and reduce the noise. Reynolds number is set to $Re = 48 \times 10^5$ and the angle of attack (AoA) was between 0° and 8° . The acoustic study indicated that the noise was reduced by 55% and 70% due to the fact that compression and reduction of vortices. Furthermore, the lift coefficient was raised while the drag coefficient was reduced by carrying out the suction and blowing.

Fatahian et al. (2019) numerically investigated the suction and blowing jets on an airfoil with and without flap through comparing the effect of single and simultaneous. In this study, it is aimed to find the most valuable flow control configuration by applying computational fluid Dynamics (CFD) method. The turbulence was imitated with SST $k-\omega$ turbulence mode and the conclusions were provided by carrying out 2-D Incompressible Unsteady Reynolds-Averaged Navier Stokes (URANS). In result, flapped airfoil was seen that it has more advantage than the without flap airfoil; such as, developed C_L/C_D proportion. They reached at the highest proportion with 73.7 for without flap model of single suction jet. The maximum separation stall angle of flapped airfoil increased from 14° to 20° , while the without flap airfoil raised by 2° .

Ohashi et al. (2019) numerically simulated the uniform blowing (UB) and uniform suction (US) to develop the performance of Clark-Y airfoil. Four different location of controller has been examined for down and upper surfaces. Reynolds number was set to $Re = 1.5 \times 10^6$ and the angle of attack (AoA) was determined as 0° . The freestream velocity UB/US was determined 0.14%. It is discovered that UB on the down region or the US on the upper area developed the C_L/C_D proportion. Briefly, it is demonstrated by the result of the study that a combined control of UB and US was the best way to develop the ratio of C_L/C_D . Specially, controlling UB on the upper front area and the US on the down rear surface were determined as the best locations.

Atzori et al. (2020) examined numerically for the effects of uniform blowing and suction by applying large-eddy simulations. NACA 4412 was used as an airfoil, for the aerodynamic efficiency to have been monitored. Moderate Reynolds number have been determined $Re = 2 \times 10^5$. They found that the applied uniform blowing decreased the efficiency, while the suction increased it. The aerodynamic efficiency increases if it carried out to the pressure side.

2.1.4. Oscillating

Kang et al. (2014) investigated numerically flow around an airfoil to control with oscillation active control method. Parametric study has been carried out on NACA 0012 with local oscillation. Aerodynamic performance was examined. In addition, frequency, equilibrium and amplitude effects of the oscillation have been carefully scrutinized. Reynolds number and the angle of attack (AoA) were set to 5×10^3 and 6° respectively. The development on lift and the flow evolutions were negotiated. The flow mold was almost unaltered compared to rigid airfoil even oscillation frequency was low, $f_{ex} = 0.41-0.7$. It caused by the weakness of the excitation on flow formation. They decided to make their study with 3-D in the future.

LeBeau et al. (2010) performed a numerical study on active control method applying oscillating at the Reynolds number $Re = 25 \times 10^3$ for NACA 4415. They aimed to compare the datas between the experimental and the data, which was provided from Particle Image Velocimetry (PIV) applying both sinusoidal and polynomial approximation to the surface oscillation. It was clearly seen that the polynomial model indicated a remarkable growth in momentum thickness prior to $x/c = 0.6$, while there was not seen in sinusoidal model. PIV results were excessively different from the simulation for both.

Strike et al. (2010) experimentally employed the wiener deconvolution method due to rebuild an unsteady surface pressure on an airfoil by a pressure sensor. From 30cm to 152cm lengths tubes were used with the diameter of 0.86 mm. Moreover, real pressure effects and inlet geometries were taken into account utilizing the pressure ports in the airfoil part. The airfoil was oscillated in pitch at 5 Hz and 10 Hz in a wind tunnel. Conclusion of the experiment showed that the wiener deconvolution method was remarkably effective the measured pressure signals to have been corrected.

2.2. Passive Flow Control

2.2.1. Partially Flexible Wings

Acikel and Genc (2018) experimentally investigated the NACA 4412 airfoil profile. The experiment have been realized by using partially flexible wing located on the leading edge of the airfoil. The wind turbine blade was between $x/c = 0.2$ and 0.7 . 3 different Reynolds number were set to $Re = 2.5 \times 10^4$, 5×10^4 , 7.5×10^4 . The proportion of C_L/C_D has been raised

approximately 100% at the angle of attack was 6° with depressing the laminar separation bubble (LSB). Exactly at this moment the Reynolds number was set to $Re = 2.5 \times 10^4$. Moreover, C_D decreased while the C_L raised.

Cleaver et al. (2014) tested experimentally the partially flexible wing that was located on the trailing edge of NACA 0012. The wings were investigated for various flexibilities and lengths. The Reynolds number set to $Re = 4 \times 10^4$. Different materials; such as, aluminum, polyster, vinyl and stainless steel, have been performed to evaluate the amount of elongation; $1/c = 0.1$, $1/c = 0.3$ and $1/c = 0.3$. The researchers acquired that the biggest development via elongation rather than clean profile has been 28%.

2.2.2. Cavity

Taherian et al. (2017) investigated experimentally the RISO airfoil. They compared the flapped and without flap RISO airfoil to thick blunt trailing edge airfoil. The Reynolds number was set to $Re = 1.5 \times 10^3$, 2×10^3 and 2.5×10^3 . The airfoil was splitted from the trailing edge. ($x/c = 0.3$) The splitted trailing edge was showed a positive effect has been demonstrated by the researchers. Moreover, due to the fact that the adding cavity on the trailing edge raised the momentum was indicated by the researchers, the increase on decrease was acquired in the wake. Accordingly to this, drag was decreased with compared to unflapped airfoil profile.

Olsman and Colonius (2011) studied numerically on NACA 0018 airfoil. They prepared a 2-D direct simulation, acquired flow imutations. Subsequently, they compared the numerical results to experimental data. The Reynolds number was set to $Re = 2 \times 10^3$ and 16 different angles of attack have been examined from 0° to 15° . They also noticed that there appeared little vortices of the shear layer oscillations around the cavity and it causes the boundary layer to be suppressed. The lift increasing was observed at 0° angle of attack because of cavity and the flow was separated at 4° and 6° by the cavity.

Lam and Leung (2018) performed a numerical study on aeroacoustic characteristics by applying a cavity on the surface of NACA 0018 airfoil. The Reynolds number was set to $Re = 2 \times 10^4$ and the angle of attack (AoA) was set to 0° . Not only the cavity castrated the sound source was demonstrated by the study, but also there appeared another acoustic source, that was made of the cavity oscillation modes. The researchers also indicated that the ratio of C_L/C_D was increased to 5.3 by the cavity.

Fatehi et al. (2019) investigated cavity on Riso_B1_18 airfoil with Reynolds number $Re=1.6 \times 10^6$. The first aim of the study was to observe the effect of cavity, where was on the surface of the airfoil. The other goal of the study was to control the stall tolerance by decreasing vortex, prevented the fluctuations. The result of the study indicates that the C_L/C_D proportion was raised via the cavity. The study was also performed experimentally due to the fact that the researchers wanted to validate the results. The proportion of C_L/C_D was raised by 31% when the angle of attack (AoA) was 14° , the increased by 57% at 20° via cavity.

Raj Mohamed et al. (2019) studied numerically on modified NACA 2412 airfoil with bioinspired nose. The study was carried out at low Reynolds number and 17 different angle of attacks, from the 0° to 16° . The goal of the study was delayed the flow separation and develop the qualification of aerodynamic. In this content, they aimed to decrease the drag and increase the lift coefficients. The maximum increment of C_L/C_D was 22.4% has been discovered. Moreover, average increment was 8.6% for the all angles of attack.

Manolesos and Voutsinas (2016) examined experimentally the LI30-FB10 flat back airfoil with a new flap device. The test was applied at the Reynolds number $Re= 1.5 \times 10^6$ and 25 different angle of attacks from -4° to 20° . They also examined different drag diminution devices. All drag diminution devices were compared and the combination of an offset cavity and flap has been decided as the best. The highest increase in aerodynamic efficiency was acquired 131%.

2.2.3. Leading Edge Slats

Bangalore et al. (1997) studied numerically with a slatted rotor. In the study, UH-60A rotor type has been used. The velocity, that was applied on the study, has been measured to with Mach number. It was determined as $Ma: 0.628$. Coefficient of lift (C_L), drag (C_D), pressure (C_P) and regular forces were indicated on the clear and the slatted rotor. The results acquired, demonstrated that the slats have advantageous on modification subordinate withdrawing dynamic stall. In addition to this, it was clearly seen that it also decreased the moment on withdrawing subordinate compared to baseline rotor.

Mishra et al. (2016) investigated numerically to the performance of slats on rotor by employing the leading edge slats. The flow velocity was measured with Mach number $Ma: 0.157$ and tip

Ma_{tip} : 0.665. The airfoil selection was made in the direction of SC 2110. Moreover, the slats were S-1 and S-6. For the S-6 model slat, thrust have been raised by 10%. The researchers noticed that the S-1 and S-6 models were pretty powerful on the performance of rotor even that in the extreme condition.

Bangalore et al. (1995) examined numerically the rotors with slats so that to demonstrate aerodynamic performance. The study was taken place with VR7 and UH-60A with clear airfoil and slatted rotors. The Reynolds number was set to $Re = 2.75 \times 10^6$. The researchers indicated that in high angles of attack, the performance of leading edge slats were affected. Furthermore, the rotor with slats, consumed less power to generate more thrust.

2.2.4. Gurney Flap

Yu et al. (2011) investigated numerically on RAE-2822 airfoil by attaching a Gurney flap due to scrutinize the effect of it. For the test was employed with the height of $h/c = 0.0025$ to 0.03 . The Reynolds number was set to $Re = 5.8 \times 10^6$. The researchers monitored that the C_L and C_L/C_D were raised at low angles of attack. Moreover, results of the study explained that the flight phases were substantially developed by the Gurney flap.

Li et al. (2007) investigated experimentally an airfoil with Gurney flap and divergent trailing edge. The Reynolds number was set to $Re = 3.15 \times 10^6$. Also, 2TM and DT5 were determined as the divergent trailing edge airfoil. 13 different angles of attack (AoA) were decided to be from -4 to 8 and the heights of Gurney flap were determined as $h/c = 0.005$, $h/c = 0.01$, $h/c = 0.015$ and $h/c = 0.02$. The Gurney flap made an increase on C_L , and the highest proportion of C_L/C_D has been raised. The researchers also noticed that rising in changing shock on the airfoil caused to increase of C_L . In the height of $h/c = 0.02$ C_L raised by 29.2% and the proportion of C_L/C_D raised by 41%. Besides that, the researchers found that the increase of C_L for the divergent trailing edge was 22.3% and 33% at the same height of the flap.

Jang et al. (1998) performed a numerical examination on NACA 4412 by adding Gurney flap due to measure the performance changes. The Reynolds number was set to $Re = 1.64 \times 10^6$ and the angles of attack was decided to be 0° , 4° , 8° , 10° , 12° , 14° , 16° and 18° . Flat plate size was decided to be as of the chord's 1-3%. The size of the Gurney flap were $h/c = 0.005$, 0.01 , 0.015 ,

0.02 and 0.03. The researchers noticed that the C_L was raised by 0.3 on the $h/c=0.015$ Gurney flap model.

Yee et al. (2007) investigated to the NACA 0012 airfoil profile by adding Gurney flap due to observe the effect of it. Aerodynamic differences was scrutinized after attaching the flap. Mach number has been conducted by changing from $Ma: 0.2$ to 0.9 . Various heights of the flap employed on the study; such as, $h/c=0.005, 0.01, 0.02$ and 0.04 . The researchers also examined the aerodynamic features of the Gurney flap. They also compared them to base airfoil with different parameters. The ideal selection was appeared as the flap height of $h/c=0.02$.

Daniel et al. (2013) examined experimentally on the aerodynamic performance of a Gurney flap on S8036 airfoil. Specifically, the Aspect Ratio (AR) has been analyzed. The Reynolds number was set to $Re= 2.5 \times 10^5$ and various angles of attack; from -6° to 28° by an increase of 2° , have been determined. The aspect ratio AR: 1, 2 and 3 have been chosen to be tested. Finally, the heights of Gurney flap were $h/c=0.01, 0.02$ and 0.04 . The researchers have discovered that an increase in flap height and a reducing in aspect ratio were requested to reach having more efficiency

Storms et al. (1994) examined experimentally to the NACA 4412 airfoil profile with Gurney flap due to the fact that to acquire the effects of the flap. For this experiment, the Reynolds number was set to $Re= 2 \times 10^6$. Several heights of the flap have been prepared to the back side of the airfoil for the experiment; for instance, $h/c: 0.005, 0.01, 0.015$ and 0.02 . The highest C_L was raised by 31% via attached the Gurney flap. Moreover, C_D was reduced the same as the lift raised.

Singh et al. (2007) examined numerically on NACA 0011 and NACA 4412 with Gurney flap due to indicate the flow around the airfoils. The Reynolds number was set to 2.2×10^6 . Moreover, the heights of the Gurney flap were set to $h/c=0.01, h/c=0.02$ and $h/c=0.03$ for NACA 0011 and $h/c=0.005, h/c=0.01, h/c=0.015$ and $h/c=0.02$ on NACA 4412 airfoil. Also, several angles of attack were tested. The researchers observed that the Gurney flap made an increase on the proportion of C_L/C_D on the NACA 0011 while there was no improvement on the NACA 4412 airfoil.

Meena et al (2018) investigated numerically to NACA 0000, NACA 0006, NACA 0012 and NACA 0018 with the Gurney flap because of scrutinizing the effect of the flap. The Reynolds number was set to 1×10^3 . Several angles of attack were tested from 0° to 20° and the heights of the Gurney flap were determined as $h/x=0.3$, $h/x=0.06$, and $h/x=0.1$. The researchers monitored that the proportion of C_L/C_D has been raised by twice after the Gurney flap to back side of the airfoil. Besides that, the coefficient of drag was raised with rising of the angle of attack. The highest proportion of C_L/C_D has raised at $h/x=0.1$ for the all airfoil compared to clean one.

2.2.5. Splitter Plate

Nakagawa (1992) examined experimentally to NACA 0018 for acquiring the effect of splitter plate where behind of a square prism. The experiment has been carried out in condition that the Mach number was set to $Ma:0.11$ and 0.91 . When both, chord and square side lengths were 20mm , there was a proportional gap calculated between prism and the airfoil as $L/D:2$. Both, shear layers and vortices were controlled by the existing of airfoil where behind of a square prism. With the result of the study, the Strouhal number was proved its independency from Mach number. Strouhal number was instantaneously increased with the shock waves becoming. Moreover, it was seen that the increase in Strouhal number caused to increase of Mach number.

Metzinger et al. (2018) tested both experimentally and numerically on FB-3500-1750 airfoil with a splitter plate that was connected to trailing edge of the airfoil. The thickness of the trailing edge was 17% and maximum thickness was 35% of the chord length. The effect of vortex shedding and attitude of splitter plate aerodynamic performance were tested. The Reynolds number was set to $Re= 6.66 \times 10^5$. The size of splitter plate was decided to be between 50% and 150% of the trailing edge length. Elongation on splitter plate length demonstrated that there was a converse effects. Moreover, the examination demonstrated that the Strouhal number increased by elongation of the splitter plate length. The best optimization for the thickness of trailing edge was determined as 50% . This optimization made an increase in drag by 27% and affected negative side on lifting by 5% .

Song (2020) numerically studied on DU97-W-300 wind turbine airfoil. A flat trailing edge and a splitter plate were attached to the airfoil. The Reynolds number was set to $Re= 3 \times 10^6$. Angle of attack was set to 4° and it was not changed. The size of the splitter plate was 10% of the

chord length. The researchers found that the splitter plate made an extraordinary effect to the wake flow and it caused to decrease drag and noise.

Ozkan (2021) investigated experimentally on the back region of a NACA 0012 by adding a splitter plate due to the fact that to understand the effect of aerodynamic performance. The Reynolds number was set to 2×10^4 and the angle of attack was decided to be 16° . Different lengths of splitter plate and chord were carried out; $L/C = 0.1, 0.2$ and 0.3 . They also compared the modified airfoil to clean airfoil. According to this study, when the splitter plate located on $L/C:0.2$, turbulent area was prevented. When the location of the plate changed to $L/C:0.3$, vortex intensity was appeared. Turbulence intensity was decreased and vortex-shedding frequency has been powerless.

2.2.6. Vortex Generators

Bragg et al. (1987) experimentally investigated a Voyager canard airfoil. There was a vortex generator applied due to understand the effect of it to aerodynamic performance. The Reynolds number was set to $Re = 9 \times 10^5$. Different angle of attack were negotiated to be examined from -4° to 18° . Also, three different vortex generator were carried out; small, large and delta vortex generators. For generating small vortexes, the vortex generator was attached to $x/c: 0.17$. For generating large and delta vortexes, the generator was located at $x/c: 0.45$. The results of the study demonstrated that the vortex generators caused to raise on C_L and decrease on C_D . The researchers suggested the delta vortex generator due to the fact that it made a high increase in lift and low raising in drag.

Kerho et al. (1993) experimentally tested on LA2573A airfoil. There was installed a vortex generator because of understanding the effect of its on aerodynamic performance. Specifically, the laminar separation bubble was scrutinized with modified airfoil by the vortex generators. The Reynolds number was set to $Re = 2 \times 10^5$ and 6×10^5 . The researchers indicated that the using of vortex generators caused to decrease C_D at low Reynolds number. Also, at the Reynolds number of $Re = 2.35 \times 10^5$, drag reduction was maximum by 38%.

Lin et al. (1994) experimentally tested an airfoil modifying with vortex generators to observe the aerodynamic effects of it. The test was applied at different angles of attack from -4° to 24° . The Reynolds number was set to $Re = 5 \times 10^6$ and 9×10^6 . The best configuration of C_L was

determined on modified with vortex generators airfoil. The vortex generator added to airfoil where at $z/c=0.0018$ for getting optimized C_L ; however the place that maximum lift acquired was negotiated to be $x/c=25$. Furthermore, the researchers observed that there was a drag issue during cruise flight due to latent vortex generators.

Seshagiri et al. (2009) examined an airfoil by modifying with vortex generators to observe the aerodynamic performance effect of it. There were six different static vortex generators used. The Reynolds number was set to $Re= 8 \times 10^4$ and $Re= 1.6 \times 10^5$. Also, the angles of attack were decided to be between -4° and 18° . The researchers observed that the vortex generators caused to increase of C_L . At the Reynolds number $Re= 1.6 \times 10^5$, the highest C_L raised by 25%.

2.2.7. Riblets and Roughness Surface

Huber et al. (1987) experimentally tested on Wortmann FX-63-137 airfoil profile by adding a trip wire roughness. The Reynolds number was set on the experiment to $Re= 1 \times 10^5$. Also, the angle of attack was determined to be between -16° and 24° . The place of the trip wire was between $0.01 < x/c < 0.03$ and the height of roughness to chord proportion was $h/c=0.00583$ and 0.003 . The researchers observed that the highest C_L as 1.54 at the angle of attack 15° .

Wang et al (2000) experimentally performed in a flat plate with riblet surface on turbulence condition. The freestream velocity was 17.5 m/s. The type of riblet used was “V” and the parameters of the experiment were height: 1mm, width: 1mm and space: 2mm. The researchers clearly seen that the riblets made an apparent positive effect on the plate compared to clean model. The turbulent intensity was raised by 6.7%. Also, the experiment demonstrated that the riblet surface made a degradation on turbulent drag.

Gopalarathnam et al. (2001) experimentally investigated three different airfoil design with applying trips. The experiment has been for different Reynolds number; $1 \times 10^5 < Re < 3 \times 10^5$. There were various combination of airfoil with the places that trips were used. Airfoil profiles were SA 7024, SA7025, SA7026. The trip places were determined to be at $x/c=0.1$, $x/c=0.2$, $x/c=0.3$. The researchers observed that the drag decreased after adding the trips on the airfoils. Moreover, the C_L increased at the Reynolds number mentioned.

Lee et al. (2005) experimentally tested a circular cylinder with applying riblets over the surface. The Reynolds number was set to $2.5 \times 10^3 < Re < 3.8 \times 10^4$. The parameters of the riblets were height; $h=180 \mu\text{m}$ and space between riblets were; $s=300 \mu\text{m}$. The researchers indicated that the C_D was decreased by 7.6% at the Reynolds number $Re= 3.6 \times 10^3$; however, C_D increased approximately 4.2% at Reynolds number $Re= 3.6 \times 10^4$.

Lee and Jang (2005) experimentally investigated to NACA 0012 airfoil profile by applying a micro riblet film on the front side of the airfoil. The test was on the condition of low Reynolds number; $1.03 \times 10^4 < Re < 5.14 \times 10^4$. The shape of riblet was “V” and the size of height was $h=176.8 \mu\text{m}$, spacing between riblets of $s=300 \mu\text{m}$ and valley of groove were $w=50 \mu\text{m}$. The researchers observed that the C_D was raised by 9.8% at the Reynolds number $Re= 4.62 \times 10^4$; however, it decreased by 6.6% at the Reynolds number $Re= 1.54 \times 10^4$.

Sareen et al. (2014) was investigated the riblet film method for drag reduction and applied to wind turbine airfoils. The researchers set the Reynolds number to $Re= 106.15 \times 10^6$. Also, three different flow regimes were tested. In terms of riblet configurations, four V-shaped riblets was carried out with the height of 44, 62, 100 and 150 μm . The riblets was located on the DU 96-W-180 to the airfoil's suction side at $x/c=0.4$ to $x/c=1$ and to the pressure side at $x/c=0.7$ to $x/c=1.0$. As a result of this study, 62- μm riblet size was the best size for all configurations. The coefficient of drag was reduced by 4.5%.

Zhang et al. (2018) performed a large eddy simulation by using riblets on an Eppler E374 airfoil. The Reynolds number was set to $Re= 2 \times 10^5$. The angle of attack was decided to be $\alpha=3^\circ$. The riblets were located at $x/c=0.3$ to $x/c=99$. The width of the riblets were $w/c=0.0025$ and a height of the riblets were $h/c=0.002$. Furthermore, the researchers attached a trip at $x/c=0.13$. Its width was decided to be $w/c=0.01$. As a result of their study, while the coefficient of lift was increased, the coefficient was decreased.

Transverse groove structure was researched by Wu et al. (2019). They decided to use NACA 0012 as an airfoil. The spacing was changing with $s=0.1, 0.2, 0.5$ and 1 mm . Also, the distance between tips of riblet and their height were equal to spacing length. The riblets were located on the airfoil as two different classes. The first class was named middle part. The location of the part was between $x/c=0.3$ and $x/c=0.7$. The second class was named posterior part and their

location was between $x/c=0.5$ and $x/c=0.9$. The researchers reported that the transverse groove caused a decreasing drag coefficient by 9.65% on $s=0.1$ mm configuration.

2.2.8. Flexible Membrane Wing

Viieru et al. (2006) scrutinized the aerodynamics of flexible wings and flaps. The Reynolds number was set to $Re=10$, 120 and 6×10^3 . The study was carried out to focus on the effects of flaps motion on the wings, leading edge vortices and movement of pick-up. At the Reynolds number $Re=10$; the researchers observed that the vortex circle was attached to leading edge vortex. Moreover, at the Reynolds number $Re=120$; they investigated that the vortex corruption was withered and leading edge vortices fused with tip vortices. Moreover, the researchers tested that at the Reynolds number $Re=10$, the vortex circle was merged with the leading edge vortex, tip vortex and trailing edge

Lian et al. (2007) studied numerically on the SD 7003 airfoil profile. In the study, the laminar to turbulent transition performance was scrutinized. The Reynolds number was set to $Re=6 \times 10^4$. Also, the angles of attack were decided to be between -5° and 15° . Latex membrane was located at $x/c=0.33$ to $x/c=0.52$ without an advance tension on the front of the airfoil. The researchers showed that the flexible wing trembled. During this process, the energy was transferred, which was released from flexible surface to the flow. The result of the study indicated that the flexible wing caused to separation and transition operations.

Altar et al. (2012) experimentally investigated on a flat plat membrane wing. The scope of experiment was the understand of the aerodynamic effects. There were three different advance tension applied to the membrane, 5, 7 and 10%. The Reynolds number was set to $Re=1.37 \times 10^5$, 2.26×10^5 and 3.63×10^5 . Also, different angles of attack were decided to be test until 27° . The researchers realized that the dependence of lift before stall point was small enough to be negligible on the advance tension model. Besides that, the researchers monitored a strong relation between the Reynolds number and average lift at the level, which was advance tension applied. There were an increase in the average of drag due to decrease in advance of tension of the model. The researchers reported that there was a weak bond between the stall angle and advance tension value and it depends on the Reynolds number.

Timpe et al. (2013) investigated experimentally on a flat plate wing. In addition, there was a base membrane used for the experiment, which was made with silicone, on trailing edge. The Reynolds number was set to $Re = 5 \times 10^4$. Two different advance tension modes were compared by the proportion of 1.3% and 2.8% on the plate. The researchers indicated that the lift bend linear together finite wing theory up to 10° angle of attack. Between 25° and 30° angle of attack, the highest lift was monitored. This result was welcomed as regular for the low aspect ratios. Moreover, the proportion of C_L/C_D was reached at the maximum value at 6° and 8° . The researchers indicated that the membrane wings were useful to increase C_L/C_D and at the minimum pre-strain 1.3%, the ratio of C_L/C_D was around 5.4.

Combes et al. (2015) studied numerically on flexible fixed wing. The study was the best to raise wing efficiency in the pre-stall condition in order to demonstrate of fluid constraction coaction. The Reynolds number was set to $Re = 7.5 \times 10^4$. The results of the study demonstrated that the resistance could be raised by 7% set against to a rigid wing.

Genc et al. (2016) investigated a flexible membrane wing. The aspect ratio was set to one. The Reynolds numbers were set to $Re = 2.5 \times 10^4$ and $Re = 5 \times 10^4$. Also, different angles of attack were decided to be such as, 12° , 16° , 20° and 24° . The results of the study showed that membrane deformation was increased by angle of attack. Moreover, the oscillation mode was reduced and the place of highest standard that deflection occurs continued heading to the trailing edge. The researchers cleanly seen that it occured on the flexible wing due to raise in the angle of attack.

2.2.9. Leading Edge Serrations

Ito (2009) experimentally investigated the leading edge serration on airfoils due to the fact that to demonstrate the effects of aerodynamic at low Reynolds number. The Reynolds numbers were set to $Re = 2.1 \times 10^4$ and 2.1×10^5 . Also, the angles of attack were decided to be between 10° and 45° . The researchers made a catagorization among the leading edges. Three different category were appeared; 16, 20 and 26 teeth/inch. They monitored a small differences in values depending on the angle of attack. They noted that for 20 and 26 teeth/inch model demonstrated better efficiency at the Reynolds number $Re = 2.1 \times 10^4$ compared to the clean model.

Furthermore, at the $Re = 2.1 \times 10^5$ the 16 teeth/inch model was not adequate to be satisfy in terms of the lift coefficient. The researchers affected from the night owl before they made the study.

Winzen et al. (2014) tested experimentally to leading edge serrations. The Reynolds numbers were set to $Re = 4 \times 10^4$ and 1.2×10^5 . Also, the angles of attack were negotiated to be between -15° and 20° . Three different models were analyzed in this study; base, silicon based serration materials and metal. The size of the serrations was 2mm and 0.3mm in diameter. The gap between two serrations was 1mm. The results of the study demonstrated that the serrations decreased the efficiency in terms of aerodynamic. Drag increased, for example, specifically in metal serrations. However, the C_L was raised by 10% for bigger than 10° angles of attack.

Narayanan et al. (2015) experimentally tested on leading edge serrations. The aim of the study was to decrease the noise. Sinusoidal shaped profiles was used for the leading edge serration. Two different objects were tested to make a comparison to distinctions in noise diminution, NACA-65 and a flat plate. The results of the study was demonstrated that an important noise diminution was monitored at moderate frequencies between 500 Hz and 8 kHz. However, at weak frequencies, the noise reduction was negligible.

Bidermann et al. (2017) studied on leading edge serration with formed sinusoidal leading edge. It was used on the noise diminution mechanism due to define the fluctuations and loyalty on turbulence severe, Reynolds number, wavelength, angle of attack and serration amplitude. The researchers studied on NACA 65(12)10 airfoil profile. The Reynolds numbers were set to be between 2.5×10^5 and 6×10^5 .

2.2.10. Micro Small Cylinder

Li et al. (2014) experimentally performed on NACA 0012 airfoil profile. In this study, rod and airfoil alignment were investigated due to demonstrate the impact of vortex construction caused on noise radiation. Also, various rods were used in diameters of D:3mm, 5mm, 8mm, 11mm, 15mm, 18mm, 21mm and 25mm. Moreover, a square rod was used with the height of H:15mm. The gap, between airfoil and rod, was decided to be between 30 and 170mm. The Reynolds numbers were set to $Re = 6.3 \times 10^3$ and $Re = 106 \times 10^3$ situated to rod diameter and $Re_C: 45 \times 10^3$ and $Re_C: 90 \times 10^3$ situated to chord length of airfoil. The researchers demonstrated that the

critical space over diameter was 3.3 and it caused the biggest decrease in terms of noise level with more than 30 dB.

Luo et al. (2017) tested on NACA 0012. In this test, the researchers scrutinized numerically the influence of micro cylinder adjacent the leading edge. The Reynolds number was set to $Re = 6 \times 10^6$. Also, a rod was mounted and the angles of attack were set to 16° and to 23° relating to chord axis. The diameter of rod and the gap between rod and airfoil were negotiated to be between D/c : 0.005 and 0.02, s/c : 0.005 and 0.035 respectively. The researchers noted that the place and the gap affected substantially to stall conditions.

Rousoulis et al. (2017) numerically performed to NACA 0012. In this study, the researchers scrutinized the influence of a rotating rod, which was mounted to the suction side of airfoil due to control the noise. The space between the airfoil and rod was a chord length of airfoil. In terms of airfoil chord length Reynolds number was set to Re_c : 4.8×10^5 and in terms of diameter of the rod Reynolds number was set to Re_D : 4.8×10^4 . It was seen that the frequency was approximately doubled compared to the natural shedding frequency. Also, the researchers noted that the noise reduction appeared because of the rotation of the rod.

2.2.11. Slotted Airfoils

Baragona et al. (2003) performed a numerical investigation on slotted NACA 63-415 airfoil profile. Also, the researchers made an experiment due to validate the data they acquired from the numerical results. They scrutinized the influence of the slot by comparing to base airfoil. The Reynolds number was set to 2×10^6 . Moreover, the angles of attack were decided to be between -6° and 15° . Six different size of gaps between the airfoil and flap were tested; g/c : 0.015, 0.02, 0.025, 0.03, 0.035, 0.04. Also, the overlap distance was negotiated to be $O/c=0.01$ and 0.02. The highest C_L was monitored 2.53 at $g/c=0.03$ and $O/c=0.02$ alignment.

Belamadi et al. (2016) numerically examined to NREL S809 airfoil profile. The researchers scrutinized that the effects of slot to the aerodynamic efficiency. The Reynolds number was set to 1×10^6 . Different parameters were taken into account; slot slope, slot width and location of the slot, $\beta=-60^\circ$, $y/c=0.01$ and $x/c=0.2$ to 0.8 respectively. The airfoil, used in this study, was used for horizontal axis wind turbine blade in general. The angles of attack were defined to be

between 15° and 20° . After compared to all configurations, the maximum C_L was around 1.5 at 20° angle of attack.

Mdouki (2017) numerically performed the effect of slot on an airfoil due to observe the difference on aerodynamic efficiency. They decided to use S809 airfoil profile for this study. The Reynolds number was set to $Re = 0.75 \times 10^5$. Also the turbulence intensity was set to 0.3%. The main aim of the study was controlling the Laminar Separation Bubble (LSB) with the slot. The researchers indicated that the simulation demonstrated the slot was effective for the configuration chosen. However, for the small angles of attack, the slot could not lead to make an enhancement for lift and drag coefficients.

Beyhaghi et al. (2018) investigated numerically on base and slotted NACA 4412 airfoil profile. The researchers also conducted an experiment due to validate the data acquired from the numerical results and they verified. The researchers aimed to develop the aerodynamic efficiency by slotting the airfoil. The Reynolds number was set to 1.6×10^6 . Also, the angles of attack were negotiated to be between 0° and 16° . Three different parameters were investigated; width of slot, angle of inlet and vertical position. At the best configuration, researchers observed that the lift coefficient was raised by 30%; however, drag was dramatically increased at the same time.

Shehata et al. (2018) numerically investigated to one of the symmetrical airfoil profile, NACA 0015. The airfoil profile is generally used on Wells turbine blade that works for Oscillating Water Column (OWC) to acquire the energy from ocean waves. The common issue about Wells turbine was early stall. Due to delay the stall, the researchers determined to slot the airfoil. They examined the effect of incline on Wells turbine's performance in this study. The researchers was indicated that the most efficiency angle for slot was 10° clockwise with 3% and 15% development in C_T , before and after stall respectively. They compared the results to 0° angle slot. It was clearly seen that the 10° angle slot results was better. Also, the aeroacoustic noise was reduced by 21.2% owing to the slot.

Ramzi et al. (2018) investigated numerically to NACA 0021 airfoil profile. The goal of the study was to control the laminar separation bubble (LSB) through two types slots, where was located to the front side of the airfoil. The Reynolds numbers were decided to be between 1×10^4

and 1×10^6 . Also, increasing angles of attack were applied. The slots were categorized into two in this study. One is transversal and the other is longitudinal slot. The width of the transversal slot was $0.005c$. (0.5% of the chord length). The slope was 45° of the slot. The researchers observed that the longitudinal slot made an increase by 4% on the lift coefficient and it caused to decrease the length of laminar separation bubble. However, it also increased the drag coefficient by 2%.

Ni et al. (2019) numerically investigated on NACA 634-021 airfoil profile. Also, an experiment was conducted to validate the data supplied from the numerical study and the data was verified. The researchers studied on the influence of slot on the airfoil. The airfoil model, used in this study, was used for wind energy turbine blades in general. The Reynolds number was set to $Re = 1 \times 10^5$. Also, the angles of attack were decided to be between 0° and 30° . Moreover, nine different slot alignments were examined to acquire the highest ratio of C_L/C_D . The researchers reported that, at the best alignment, the highest C_L was raised by 58%. Furthermore, the highest proportion of C_L/C_D was increased by 14%.

Tang et al. (2019) examined numerically to blade-end slots. The researchers, also, conducted an experiment due to the fact that to validate the data, which were supplied from the numerical study and they verified. The freestream velocity was measured as $Ma: 0.59$. It was reported that in terms of the total pressure loss was substantially diminished within the range of 18.4% for -1.69° , 20.6% for 0° , 24.3% for 2° and 39.4% for 4° . It was understood from the results that the blade-end slots could produce self-adaptive high-momentum jet flows via pressure gradient. The researchers, also reported that the place, which unseparated flow occurred in the midspan of blade, was enlarged. It caused to a development on flow capacity.

Mohamed et al. (2020) numerically examined to NACA 0018. The airfoil was slotted from different locations and slopes due to the fact that to understand the influence of it to aerodynamical efficiency. The Reynolds number was set to 2.5×10^5 . Also, the angles of attack were changed from 0° to 30° . Ten different configurations were examined to acquire the best proportion of C_L/C_D . The parameters that were used in the study was; the proportion of W/C between 0.02 and 0.035, L/C was between 0.15 and 0.35 and two different slopes were applied $\beta = 45^\circ$ to 60° . The researchers noted that the slot delayed boundary layer separation to postpone stall angle, specifically at high angles of attack.

Kumar et al. (2020) numerically investigated on to E387 airfoil profile. The researchers aimed to examine the influence of passive control configurations on the airfoil. It was clearly seen that the lift and drag was affected from the thickness, inlet position and orientation of flow. The Reynolds number was set to $Re = 1 \times 10^5$. Also, the angle of attack was set to 0° . It was observed that the C_D was diminished when slot slope was set to 30° . There was always a loss for any configuration of slot position in terms of the lift coefficient. In addition to this, the maximum lift was supplied with the 30° slope of slot.

Acarer (2020), numerically performed on DU12W262 airfoil profile. The airfoil was slotted from different locations and sizes; front side and trailing edge side. The data, which was supplied from the results of the numerical study, was verified by experimental data. The Reynolds number was set to $Re = 1 \times 10^6$. C_L , C_D and the proportion of C_L/C_D was compared at the range of $-20^\circ < \alpha < 20^\circ$. The implications were numerically showed on Horizontal- and Vertical-Axis Wind Turbines (HAWT and VAWT). Results of the study showed that the C_p was raised by 3.2% for HAWT, while there was no change on C_p for VAWT.

Nia et al. (2021) numerically investigated on the flow optimization methods by using an airfoil. The researchers considered to slot the airfoil from different location. They also negotiated about carrying out different angles of attack. All the data, that were supplied from the numerical study, were verified by experimental data. The angles of attack were decided to be from 10.93° to 28.93° . The Reynolds number was set to $Re = 2.45 \times 10^5$. The researchers were reported that the location of slot becomes insignificant when the angle of attack was rising; however, in low angles of attack, the location of slot was still important. It was also noted that when the slot location was between $0.5c$ and $0.8c$, the performance of the blade was better than clean profile for both high and low angles of attack.

Fawzi and Ali (2021) numerically investigated the effect of slot on NACA 4412 airfoil profile. Three different parameters were scrutinized in the study; slot exit location, width and slope. The Reynolds number was set to $Re = 1 \times 10^6$. Also, the angles of attack were decided to be between 0° and 20° . The researchers noted that the most appropriate result was acquired when using $k-\omega$ SST turbulence model on ANSYS. The result of this study indicated that C_L was higher and C_D was lower compared to the base airfoil. Furthermore, the stall angle was postponed to 19° due to the slot. At the 18° angle of attack, the proportion of C_L/C_D raised by

110%. The researchers reported that the best results were supplied when the slot was located at $0.6c$, 65° slope and $0.1c$ width.

Zhu et al. (2021) numerically examined E387 airfoil profile with slotting from both downstream and upstream. The Reynolds number was set to $Re = 10^5$. Also, three different geometrical parameters were conducted on airfoil shape; distance of the slot's center from the center of airfoil (h_c), the slope of slot (β) and the width of slot (h_w). The highest efficiency was raised by around 6% for the generator, which was optimized slot airfoil. ($\beta: 90^\circ$, $h_c: 0.2b$, $h_w: 0.015c$) The researchers reported that the sequence of the parameters of the slot effect were $\beta > h_c > h_w$.

Santra and Greenblatt (2021) numerically investigated two different airfoil profiles; NACA 0012 and NACA 0018. Both airfoil profiles slotted and compared to their clean model. The Reynolds number was set to 2.4×10^5 . Also the angles of attack were decided to be between 0° and 20° . The researchers reported that the slots did not made an increase in lift coefficient in pre-stall angles of attack, while the good results were taken in post-stall angles of attack for both airfoils.

3. MATERIALS AND METHODS

Computational fluid analysis enable to obtain faster solution for resercher. Due to the fact that preparing an experiment could be expensive and needed to spend more time, scientists orientated to numerical methods. Also, in recent years the numerical studies on fluid dynamics increased because of the faster results received. In the present study, 2-D numerical analysis was carried out for NACA 0018 airfoil by using ANSYS Fluent computational fluid dynamics (CFD) package program. Furthermore, the numerical solutions of flow around NACA 0018 were performed using k- ω SST and k- ϵ RNG turbulence models. In this study, the Reynolds number is set and kept constant to $Re=3 \times 10^5$.

$$Re = \frac{\rho V_{\infty} L}{\mu} \quad (1)$$

In this formula, ρ refers to density of the fluid, V_{∞} indicates velocity of the freestream flow, L is characteristic length and μ refers to dynamic viscosity of fluid. In the field of aerodynamic, the characteristic length in Reynolds number formula is taken as chord length of airfoil, c . The magnitude of the forces on a wing varies with parameters such as the density of the fluid, viscosity, the surface area of the wing and the chord length. For this reason, it is more useful using non-dimensional numbers. Several dimensionless parameters have been reproduced in order to have an idea about the performance of the wings such as drag coefficient (C_D), pressure coefficient (C_p) lift coefficient (C_L).

$$C_p = \frac{p - p_{\infty}}{\frac{1}{2} \rho V_{\infty}^2} \quad (2)$$

The pressure distribution around the airfoil can be dimensionless by using Eq.2 and the dimensionless number is called as the pressure coefficient (C_p). In this equation; p is the static pressure of the measuring point on the body, p_{∞} is the static pressure of the freestream flow, v_{∞} is the velocity of the freestream flow, and ρ is the density of the fluid. The lift, drag and moment coefficients can be obtained by using pressure coefficient, as well as the location of the laminar separation bubble formed on the wing surfaces, the separation and re-attachment locations can be determined.

$$C_L = \frac{F_L}{\frac{1}{2} \rho V_{\infty}^2 A} \quad (3)$$

$$C_D = \frac{F_D}{\frac{1}{2} \rho V_\infty^2 A} \quad (4)$$

Lift and drag coefficients are calculated by using Eq.3 and Eq.4, respectively. In the equations, F_L and F_D are lift and drag forces, respectively and A is reference area.

3.1. Model and Computational Grid

In this study, the NACA 0018 airfoil profile was numerically analyzed for different parameters. The chord length taken as 150 mm. The origin of coordinate system was placed at the trailing edge of airfoil. The C-type mesh structure was used for providing a more uniform distribution around the airfoil and it has better prediction of boundary layer flows. The size of flow domain is shown in the Figure 3.1. The fluid domain was extended 12.5c and 25c in the y-axis and x-axis directions respectively so that the wall effect of preventing the disruption of the flow structure. The mesh structure was generated by ICEM CFD. There was 105 000 mesh cells were generated around NACA 0018 profile. When generating mesh structure, y^+ value was set as to be lower than 1. The generation of mesh over NACA 0018 was illustrated in the Figure 3.2. The surface of the airfoil was divided by 120 nodes for the suction surface and 200 nodes for the pressure surface. The mesh quality table was presented in the Table 3.1.

Table 3.1. Mesh Quality Table

	Min	Max	Average	Fluent User Guide
Skewness quality	1.3E-10	0.13	0.024	Max.Skewness<0.25, Excellent
Orthogonal quality	0.75	1	0.88	0.7<Min.Orthogonality<0.95, Very Good

In order to make a better prediction for lift and drag forces, the first layer thickness was set to 7.1×10^{-5} m. Applied mesh structure in this study is shown in Figure 3.2.

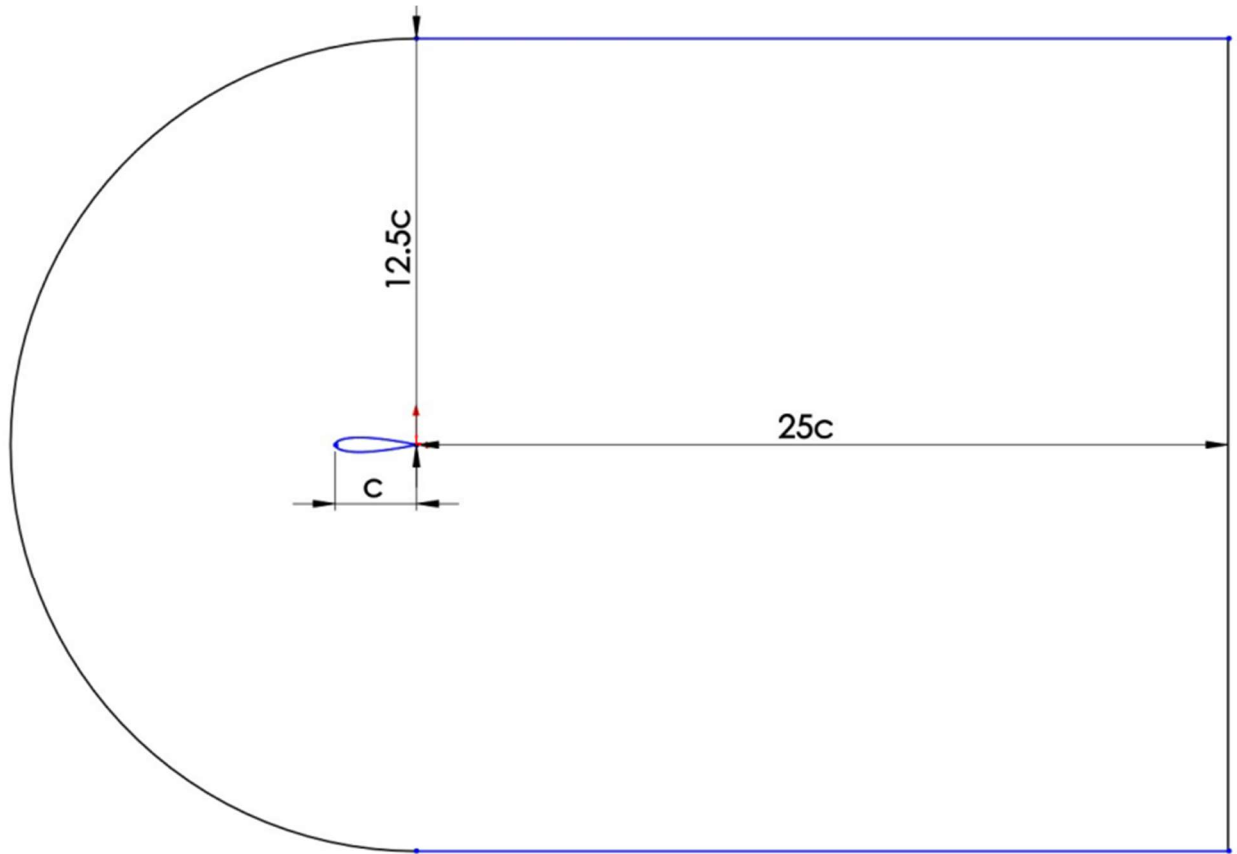


Figure 3.1. The View of Flow Domain

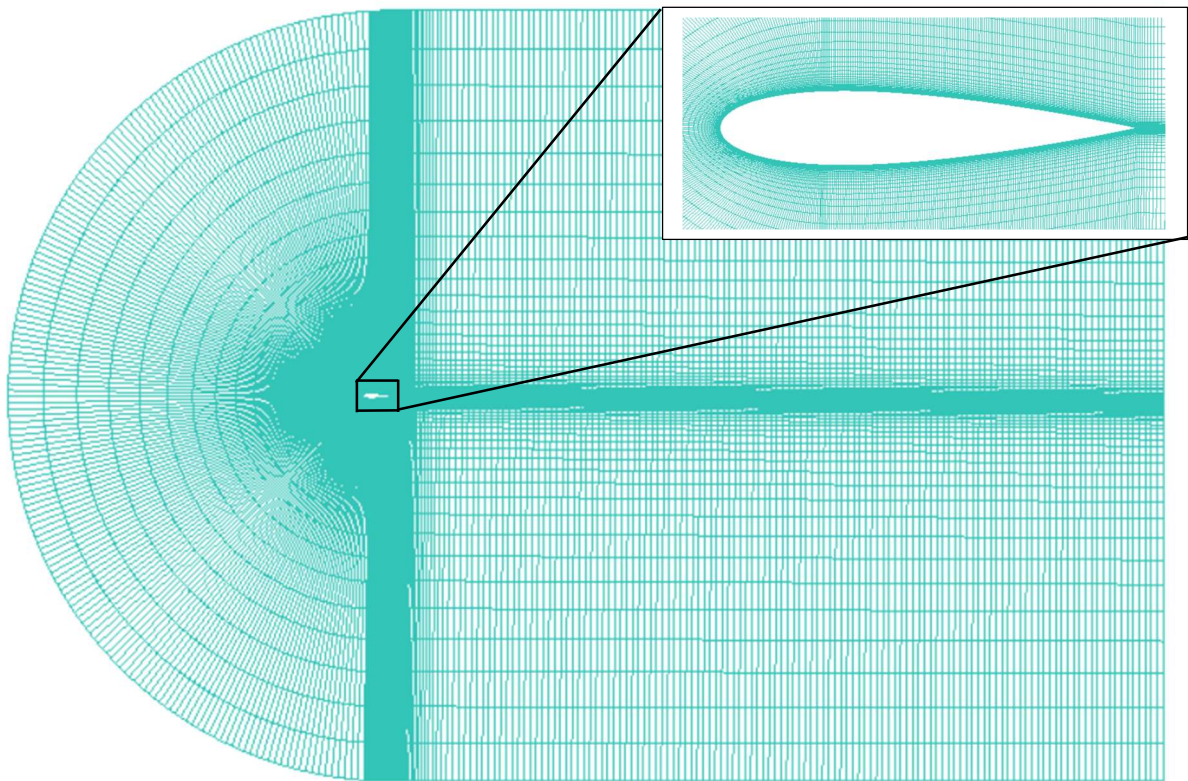


Figure 3.2. Mesh Structure around base airfoil

In this study, the aerodynamic performance of NACA 0018 airfoil is targeted to be increased by using slot. The parameters of the study is presented in Figure 3.3. In the Figure 3.3, C indicates the length of chord, W_1 denotes the slot outlet width, W_2 denotes the slot inlet width and R indicates the radius of the slot inlet and outlet. The distance between leading edge and slot outlet is designated as $L1$ and $L2$ indicates distance between the midpoint of the slot's inlet and the leading edge. A previous study on the slotted airfoil was performed by Mohamed et al. (2019). They examined the effect of slot on Darrieus type vertical axis wind turbine (VAWT). Their study was performed numerically and Realizable $k-\epsilon$ model, the shear stress transport $k-\omega$ SST model and $k-\epsilon$ RNG model were used as turbulence model. The researchers were used a NACA 0018 airfoil profile to improve the ratio of C_L/C_D and they also aimed to prevent or delay the flow separation. They set the Reynolds number to $Re = 2.5 \times 10^5$. The parameters of their study were; location of the midpoint of the slot from the leading edge (L), outlet width of the slot (W_1), inlet width of the slot (W_2), inclination angle of the slot (β). In this study, while the inlet position of the slot was kept constant ($x/c=5\%$), the outlet position was changed ($x/c=20\%$, 40% and 60%). Thus, the importance of the outlet location of the slot according to different angles of attack was investigated.

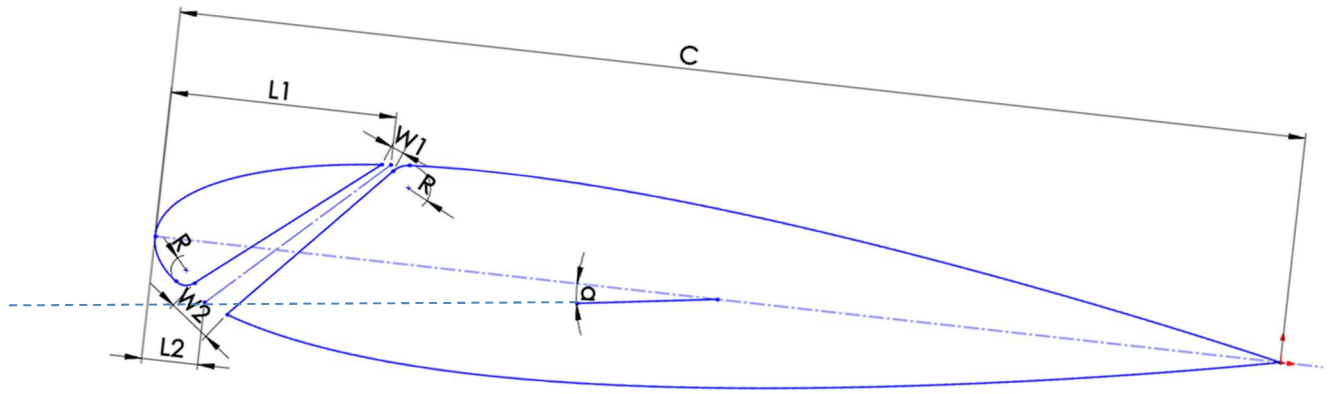


Figure 3.3. The parameters of the Study

Table 3.2 illustrates a slotted airfoil with the parameters location (x/c and $L2/c$), outlet of the slot width ($W1$), inlet of the slot width ($W2$), curves for both inlet and outlet of slot edges (R) and the angle of attack (α). Tested parameters are given in Table 3.2 and all parameters nondimensionalized with chord length of airfoil. Model C demonstrates base NACA 0018

profile. Model 20 indicates that the slot outlet is located at 20% of chord wise on the upper side of the airfoil as shown in Figure 3.5. Similarly, slot outlet is posinated at 40% and 60% of chord wise for Model 40 and Model 60, respectively. A schematic representation of Model 40 and Model 60 is presented in Figure 3.6 and 3.7, respectively.

Table 3.2. Test Cases of Slotted Airfoil

CONFIGURATION	Model 20	Model 40	Model 60	Model C
W1/C	1%	1%	1%	-
L1/C	20%	40%	60%	-
L2/C	5%	5%	5%	-
W2/W1	4	4	4	-
R/C	2%	7%	18%	-

The ratio of W2/W1 was 4.0 as recommended from Belamadi et al. (2016). Due to prevent jet flow effects, both slot inlet and outlet edges must be curved. Especially the curve of the outlet part should be adjusted properly. Curving the outlet of slot is the one of the most important process of the study in order to increase the momentum of the boundary layer. If the curve is not been carried out, the flow separation leads to decrease in the lift force.

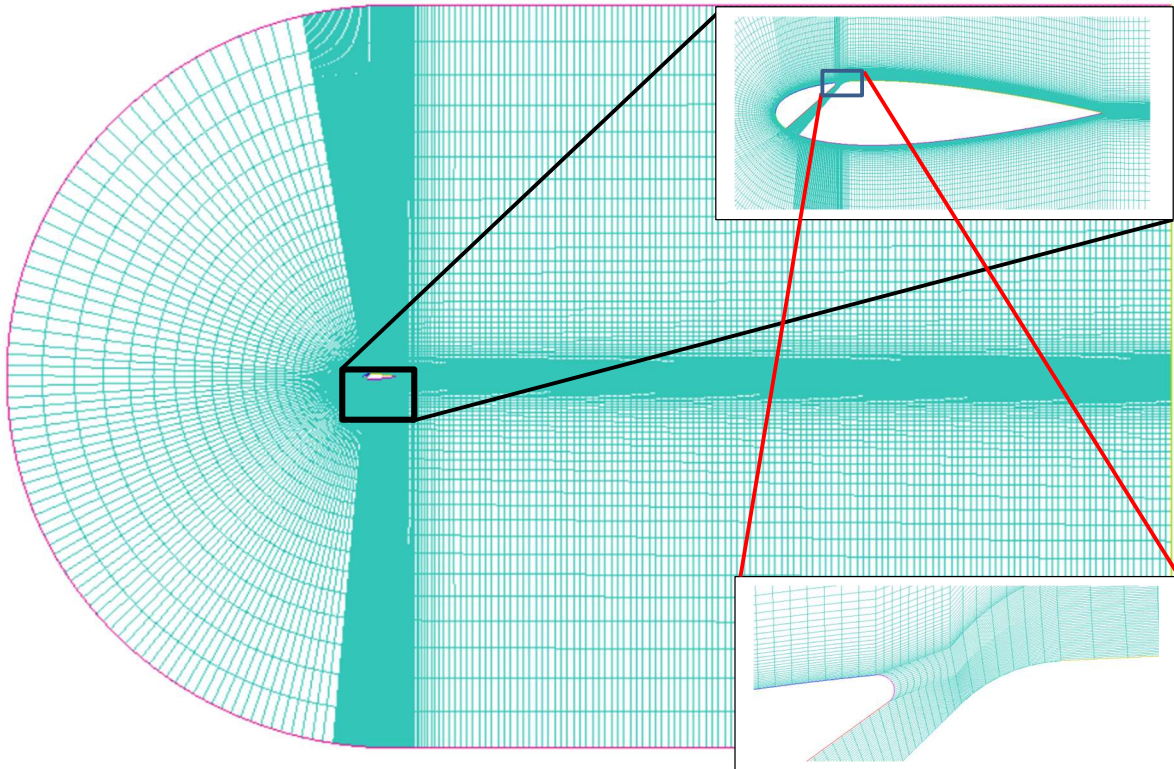


Figure 3.4. Mesh Generation of Slotted Airfoil

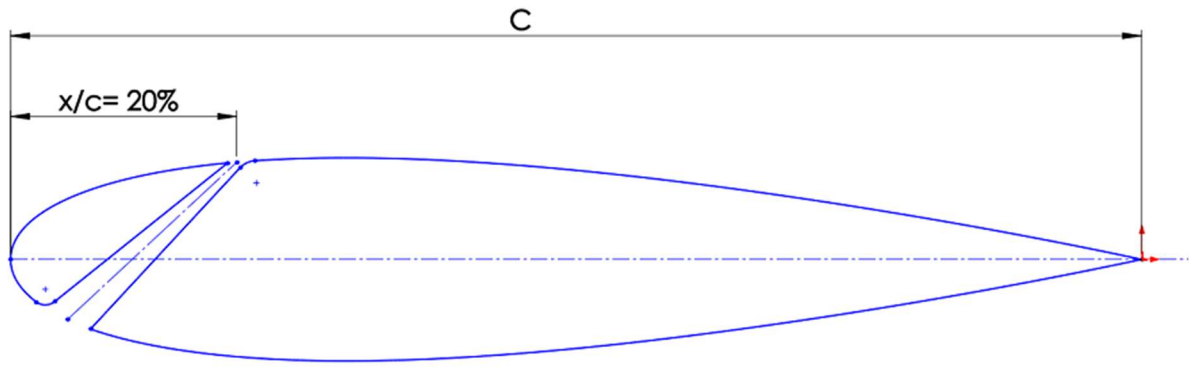


Figure 3.5. Sketching of Model 20

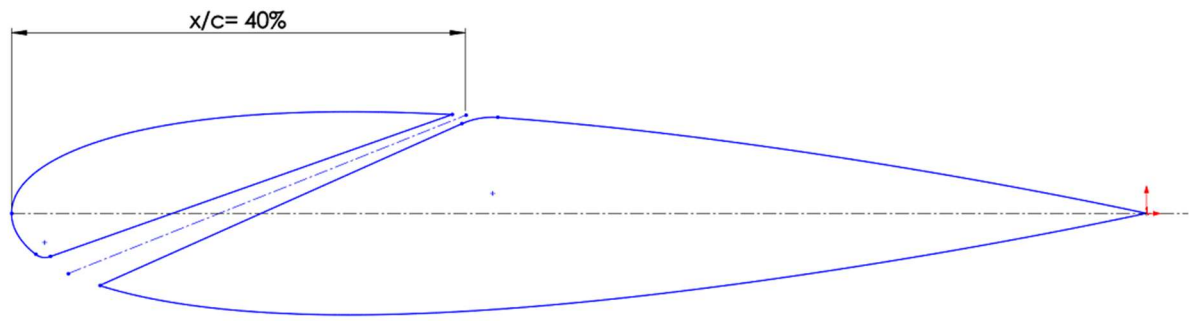


Figure 3.6. Sketching of Model 40

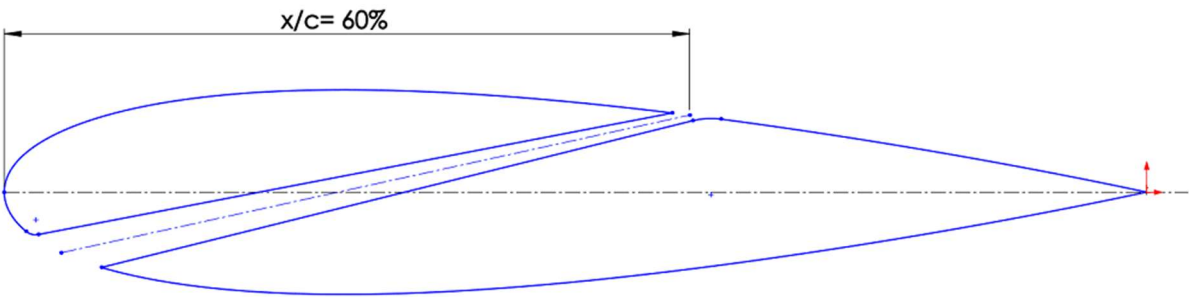


Figure 3.7. Sketching of Model 60

The slot inlet location (L2) was located at 5% of the chord length and lower side of the airfoil. This location was determined by taking into consideration previous studies. In the present study, the effect of outlet slot location was examined at different angles of attack ($\alpha=0^\circ, 4^\circ, 6^\circ, 8^\circ, 10^\circ, 12^\circ, 13^\circ, 14^\circ, 15^\circ, 16^\circ, 17^\circ, 18^\circ, 19^\circ$ and 20°). The Reynolds number was set to 3×10^5 . The mesh generation of slotted airfoil is shown by Figure 3.4. Total of 200 000 cells were generated. The mesh element size was formed as smaller towards the airfoil due to get more trustable and correct results. As it is mentioned earlier that the y^+ value set to less than 1 and the growth rate was set as 1.2.

3.2. Numerical Model and Boundary Conditions

Due to get results for the problem, in this study, ANSYS Fluent is used. ANSYS Fluent is based on RANS turbulence models (Reynolds Averaged Navier Stokes) provided by continuity, momentum and energy equations. The continuity, momentum and energy equations are given in Eq.5, 6 and 7 respectively. In these equations, $\bar{\tau}$ represent to stress tensor, $\rho \vec{g}$ gravitational force, ρ density and $\vec{F}$ represent to body forces. On the Eq.8; k_{eff} represent to effective conductivity, $\vec{J}_j$ represent to diffusion flux of species. The right side of the energy equality exemplify two facts. One of them is energy transfer that is caused by conduction another one is species diffusion and viscous spreading. Moreover, S_h occurs of chemical reaction and other volumetric heat sources. (Fluent, 2013)

$$\frac{\partial p}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (5)$$

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla_p + \nabla \cdot (\bar{\tau}) + \rho \vec{g} + \vec{F} \quad (6)$$

$$\frac{\partial}{\partial t} ((\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot (k_{eff} \nabla T - \sum h_j \vec{J}_j + \tau_{eff} \cdot \vec{v})) + S_h \quad (7)$$

The numerical analyzes were applied with 2-D, density based, implicit formulation. In this study, two different turbulence models were used to solve the problem. While determining these turbulence models, studies in the literature were taken into account. Mohamed et al. (2019) showed that the closest numerical results to the experimental data were found by k- ϵ RNG turbulence model. In this study, the results obtained from K- ω SST model were compared with the k- ϵ RNG due to observe the differences. The k- ϵ RNG model was founded by Yakhot et al. to analysis the Navier Stokes equations due to calculate for the effects of smaller motion (Eq.8 and Eq.9). On the other hand, the k- ω SST model was founded by Menter. K- ω SST is the model that has two equation large eddy-viscosity model. (Eq.10 and Eq.11) These equations contain the transport equations to represent the turbulent properties of the flow. Therefore, the K- ω SST model is more eligible to use for Low Reynolds number flow without an extra function.

$$\rho \cdot \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \epsilon - Y_M \quad (8)$$

$$\frac{D(\rho\varepsilon)}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon 1} \frac{\varepsilon}{k} G_k - \rho C_{\varepsilon 2} \frac{\varepsilon^2}{k} \quad (9)$$

In these formulas; k represents for kinetic energy, ε represents for the rate of dissipation. G_k represent to the generation of turbulent kinetic energy (TKE) that occurs from the velocity gradients. G_b represent to generation of TKE that comes from buoyancy. Y_M represent to the contribution of the fluctuating expansion in compressible turbulence and ω represent to the specific dissipation rate.

$$\rho \cdot \frac{Dk}{Dt} = \tau_{ij} \frac{\partial \bar{U}_i}{\partial x_j} - \rho B^* k \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (10)$$

$$\rho \cdot \frac{D\omega}{Dt} = \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial \bar{U}_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + 2\rho(1 - F_1)\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (11)$$

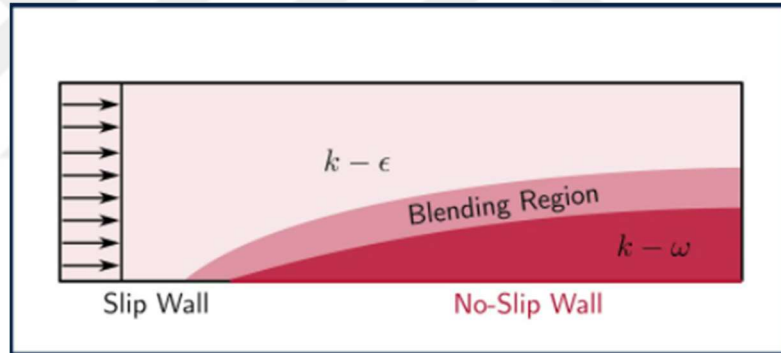


Figure 3.8. Region that Separates $k-\omega$ from $k-\varepsilon$ (Fluid Mechanics 101, 2021)

All solutions were operated at steady-state solution with using coupled-implicit algorithm scheme was operated for density-based solver. For gradient, the least square cell-based method was used. For spatial discretization, second order upwind method was utilized. No-slip wall boundary condition was applied for airfoil. Also, the flow region was chosen as pressure far-field condition by 5% of turbulent intensity and 0.1992m of hydraulic diameter. Residuals for parameters were set to 10^{-8} convergence absolute criteria.

3.3. Validation

First of all, mesh independence study was performed in order to obtain sensitive results from the numerical solution. At the angle of attack of $\alpha=10^\circ$, five different element numbers were examined for base airfoil. Obtained lift coefficients were compared with the results of Shabur

et al.(2021), Jacob (1937) and Timmer (2008). In this study, the validation was made with K- ω SST turbulence model.

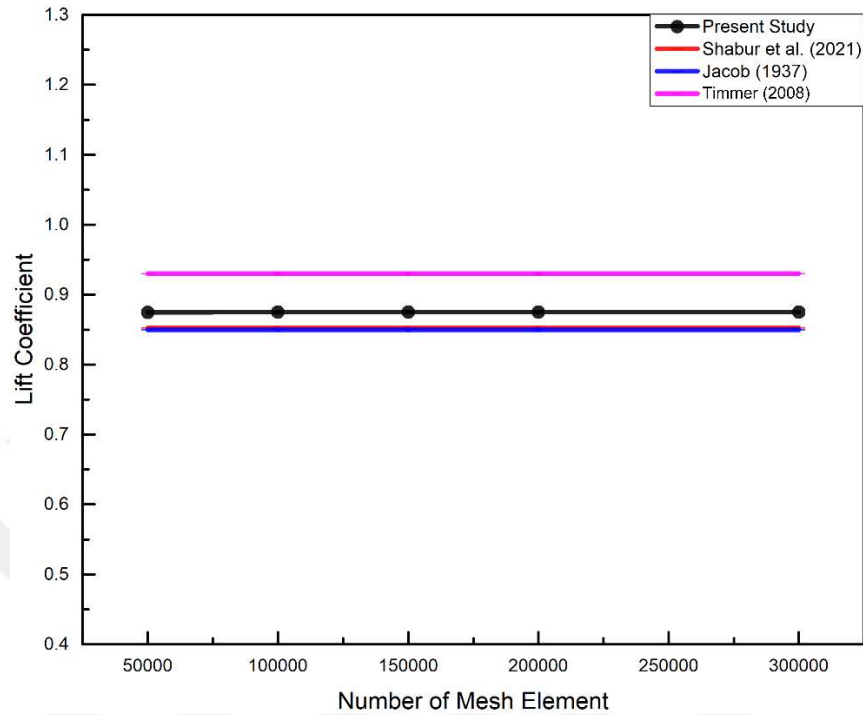


Figure 3.9. Mesh Independence Study

Figure 3.9 illustrates that the variation of C_L with the number of mesh element at the angle of attack $\alpha=10^\circ$. It is clearly seen from the figure that the increase in the number of mesh element does not affect lift coefficient, significantly. In the present study, the number of mesh element was taken to be 100 000 since the C_L was nearly the same with the experimental results. Furthermore, the values of C_L were compared with experimental results, obtained from literature, for several angles of attack; $\alpha=6^\circ$, $\alpha=10^\circ$, $\alpha=12^\circ$, $\alpha=15^\circ$, $\alpha=16^\circ$, $\alpha=18^\circ$ and $\alpha=20^\circ$ at the Reynolds number of $Re=3 \times 10^5$.

Figure 3.10 shows the variation of lift coefficient (C_L) according to angle of attack. It is clearly seen that the value of C_L value in our study has a good overlap on the curve of the C_L that is found from experimental studies in the literature.

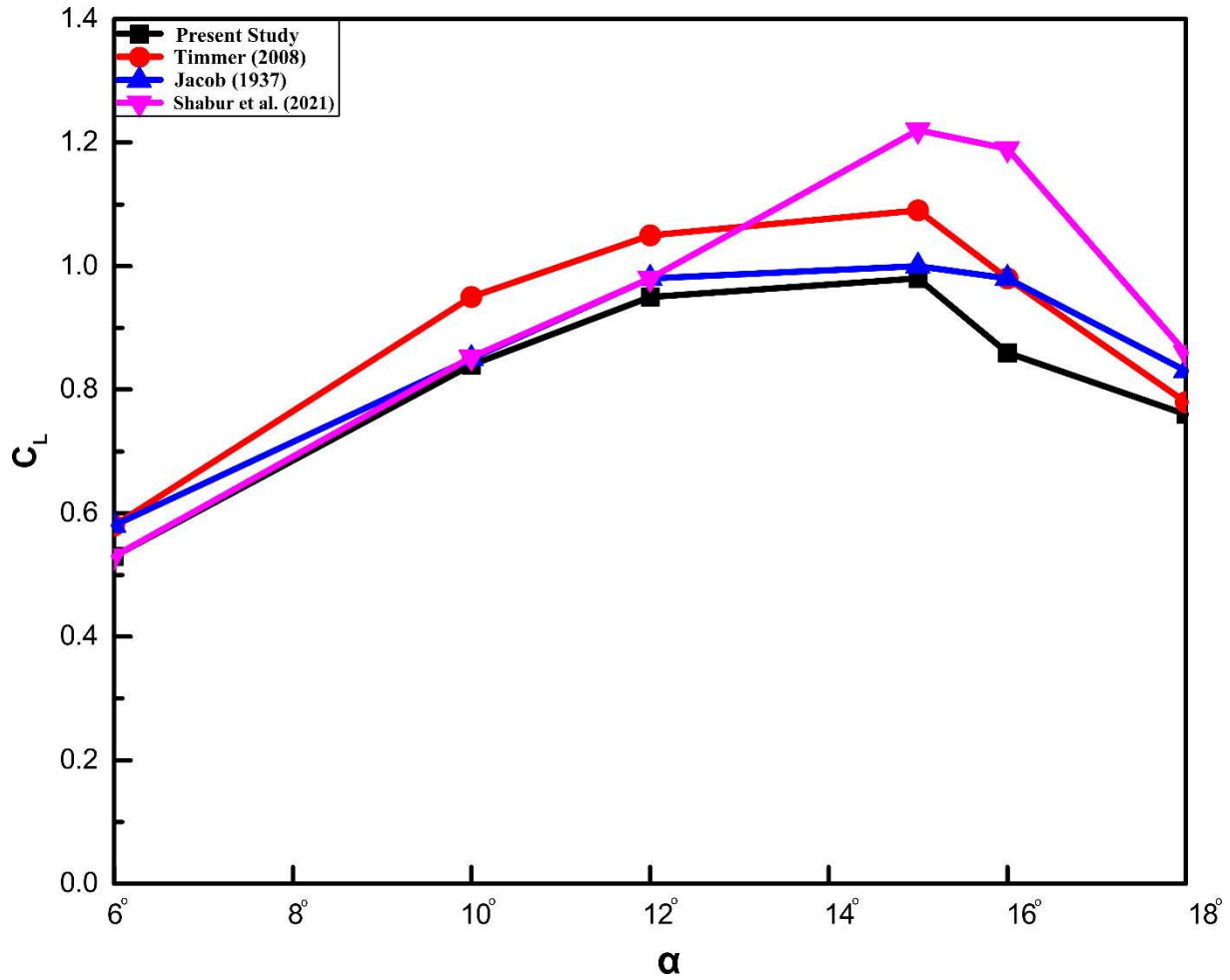


Figure 3.10. Comparison of Experimental Results and Present Results

A validation was also made for the slotted airfoil. The case 9 model used by Mohamed et al (2019) in their study was drawn in the solidworks program according to the indicated parameters. In the study of Mohamed et al (2019), numerical analysis was performed at $Re=2.5 \times 10^5$ using pressure based. In this study, with the same regulations; however, a density based analysis was made. Two of the turbulence models recommended by Mohamed et al were used. Based on the results of these two turbulence models for 10 and 15 angles of attack, Table 3.3 was prepared and the slotted airfoil was confirmed with the results in the literature.

Table 3.3. Validation of Slotted Airfoil by comparison to the literature

	Mohammed et al. (2019)	k- ω SST	k- ϵ RNG
AoA	CL	CL	CL
10°	0.85	0.75	0.83
15°	1.3	1.23	1.31

4. RESULTS AND DISCUSSIONS

In this study, it was targeted to control the flow around NACA 0018 airfoil, with creating slot at different location ($x/c=20\%$, 40% , 60%). Firstly, it is needed to determine the flow patterns that occur around the base airfoil in order to constitute reference and make a comparison. Due to this reason, unmodified airfoil profile was tested at the Reynolds number of $Re=3 \times 10^5$ for different angle of attack from 0° to 20° . There are two turbulence model have been applied ($k-\omega$ SST and $k-\epsilon$ RNG) to solve the problem. Obtained results from the numerical solution was also compared with the experimental results. The comparisons of obtained from turbulence models and experimental results at different angle of attack were shown in Figure 4.1. It is seen from the graph that the $k-\omega$ SST model was more compatible with the experimental results. While this turbulence model makes good predictions at $0^\circ \leq \alpha \leq 6^\circ$, error rate increases at $6^\circ < \alpha \leq 10^\circ$. However, it is observed that the error rate decreases at $\alpha \geq 10^\circ$. Especially for $\alpha = 17^\circ$, 18° , 19° and 20° , it is observed that obtained results almost the same with the experimental result. The highest error of this model is obtained at $\alpha = 10^\circ$ which is approximately 12%. On the other hand, obtained results from $k-\epsilon$ RNG turbulence model are much better than the $k-\omega$ SST turbulence model up to angle of attack of $\alpha = 13^\circ$. However, it is observed that obtained lift coefficient values and stall angle are higher than experimental results for $\alpha \geq 13^\circ$.

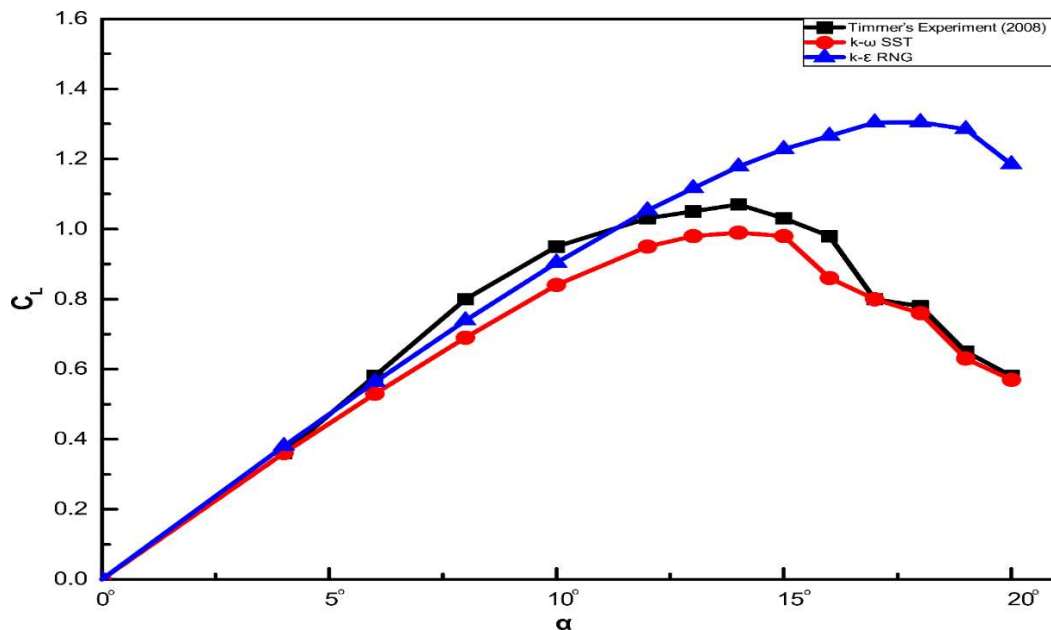


Figure 4.1. Comparison of turbulence models with base airfoil at different angle of attack

4.1. K- ω SST Model

In this section, the effect of slot on the flow characteristics around airfoil was investigated using k- ω SST turbulence model. Figure 4.2 shows that the variation of the lift coefficient with the angle of attack for different slotted configuration. For base airfoil, it is observed that the lift coefficient is zero at angle of attack of $\alpha=0^\circ$ since NACA 0018 airfoil has symmetrical geometry. On the other hand, it is observed that the value of lift coefficient increases with the increasing angle of attack. The maximum value of lift coefficient is determined as $C_L=0.99$ at the angle of attack of $\alpha=14^\circ$. It is clearly seen that a sudden decrease in the lift coefficient to $C_L=0.86$ was observed at the angle of attack of $\alpha=16^\circ$ due to the stall condition. For the slotted models, lift coefficients are below zero at angle of attack of $\alpha=0^\circ$. The reason for this is that it has turned into an airfoil with impaired aerodynamics. However, the value of lift coefficient increases as the angle of attack increases. All models could not achieve a greater lift coefficient than the base airfoil up to the angle of attack of $\alpha=6^\circ$. It is observed that Model 40 is more successful after $\alpha=15^\circ$. It is found that stall angle is delayed up to 4° for Model 40. The stall angle for base airfoil is determined as $\alpha=16^\circ$, while it is determined as $\alpha=20^\circ$ for Model 40. The maximum lift coefficient is obtained as 1.6 at angle of attack of $\alpha=19^\circ$ for Model 40. In other words, the maximum lift coefficient increases by about 154% when compared to the base lift coefficient of airfoil at 14° . It is observed that the lift coefficients of Model 20 up to angle of attack of $\alpha=13^\circ$ are lower than all other models. However, stall angle is delayed in comparison with the base airfoil. The highest stall angle is obtained for Model 20, although obtained lift coefficients are lower than other slotted models. It is seen that Model 20 is more successful than the Model 60 at $\alpha=18^\circ$. In addition, it is observed that the C_L value of Model 20 at $\alpha=20^\circ$ is higher than that of Model 40 since Model 40 is already stall. It is observed that Model 60 has the highest lift coefficient among the controlled models at low angles of attack and while it has lower than the base airfoil. While the C_L value is obtained as 1.32 at the angle of attack of $\alpha=14^\circ$, it decreased to 1.31 at the angle of attack of $\alpha=15^\circ$. However, the C_L value is increased by 10% to approximately 1.44 at the angle of attack of $\alpha=16^\circ$. Model 60 stalls at the angle of attack of $\alpha=19^\circ$. That is, the stall angle is delayed 3° relative to the base airfoil.

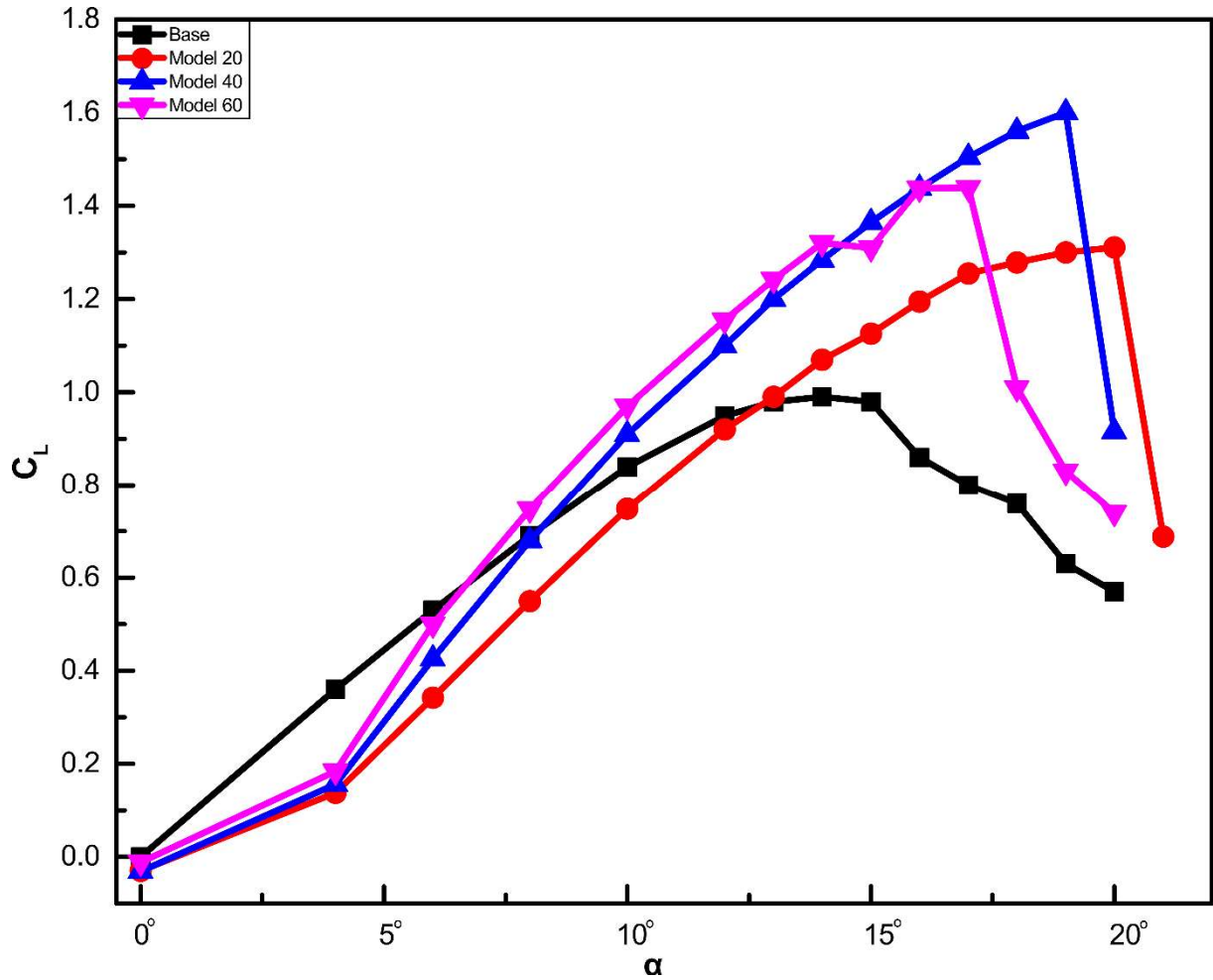


Figure 4.2. Variation of lift Coefficient with the angles of Attack for different models

Figure 4.3 shows the drag coefficients corresponding to the angles of attack for different models. For the base airfoil, the drag coefficient is calculated as $C_D = 0.017$ at zero angle of attack. As the angle of attack increases, the front cross-sectional area of the airfoil also increases. Therefore, there is a direct relationship between the angle of attack and the drag force. At angles of attack of $\alpha \leq 10^\circ$, the drag coefficient increases very slowly; When the angle of attack is $\alpha = 16^\circ$ and the stall situation occurs, the drag force suddenly increases by about 200% (while increasing from $\alpha = 14^\circ$ to $\alpha = 17^\circ$). In other words, with the increase in angle of attack causes a sharp increase in drag coefficient. Maximum drag is observed as $C_D = 0.0208$ at $\alpha = 20^\circ$. Because of slot, until the angle of attack of $\alpha = 8^\circ$ in controlled airfoils, higher drag coefficient is observed when compared to the base airfoil. The drag coefficient increases 10-15% up to $\alpha = 14^\circ$, while it significantly increases by 25-30% and more as from $\alpha = 15^\circ$. That is caused by stall. By looking at the other models, while the percentage of drag increase is not very high up

to $\alpha=18^\circ$, in this context, the Model 60 show the sudden increase at $\alpha=18^\circ$. Among all models, the model with the slowest increase in drag coefficient is found as Model 40.

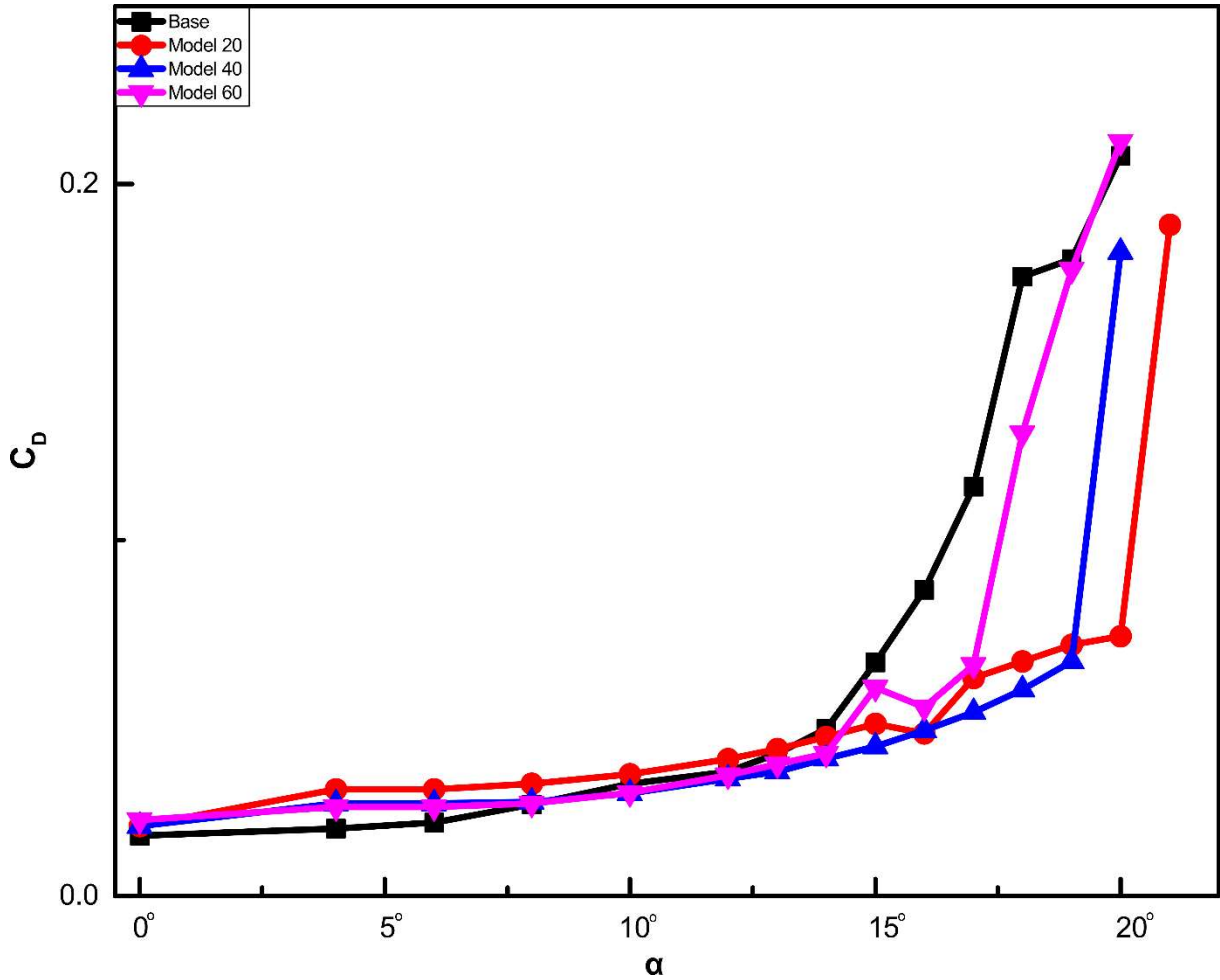


Figure 4.3. Variation of drag Coefficient with the angles of Attack for different models

Figure 4.4 shows the ratio of the lift to drag with the angle of attack for the base airfoil and all the controlled models. It is clearly seen that the controlled models have a lower ratio than the base airfoil at angle of attack of $0^\circ \leq \alpha \leq 8^\circ$. For the base airfoil, this ratio is almost constant at the angles of attack between $5^\circ \leq \alpha \leq 12^\circ$, while the Model 40 and Model 60 increase with the angle of attack. The Model 20 has a lower ratio than the base airfoil in these range of angles of attack. At $\alpha=12^\circ$, the value of base airfoil is 27.14, while they are evaluated as 23.89, 33.3, and 33.97 for Model 20, Model 40 and Model 60, respectively. The highest C_L/C_D ratio (which is approximately 34.28) is obtained for Model 40 at 13° . This value is 26% higher than that obtained highest value from base airfoil at $\alpha=12^\circ$. While the performance of the base airfoil decreases rapidly at $\alpha \geq 13^\circ$, the ratio of lift to drag slightly decrease for controlled airfoils.

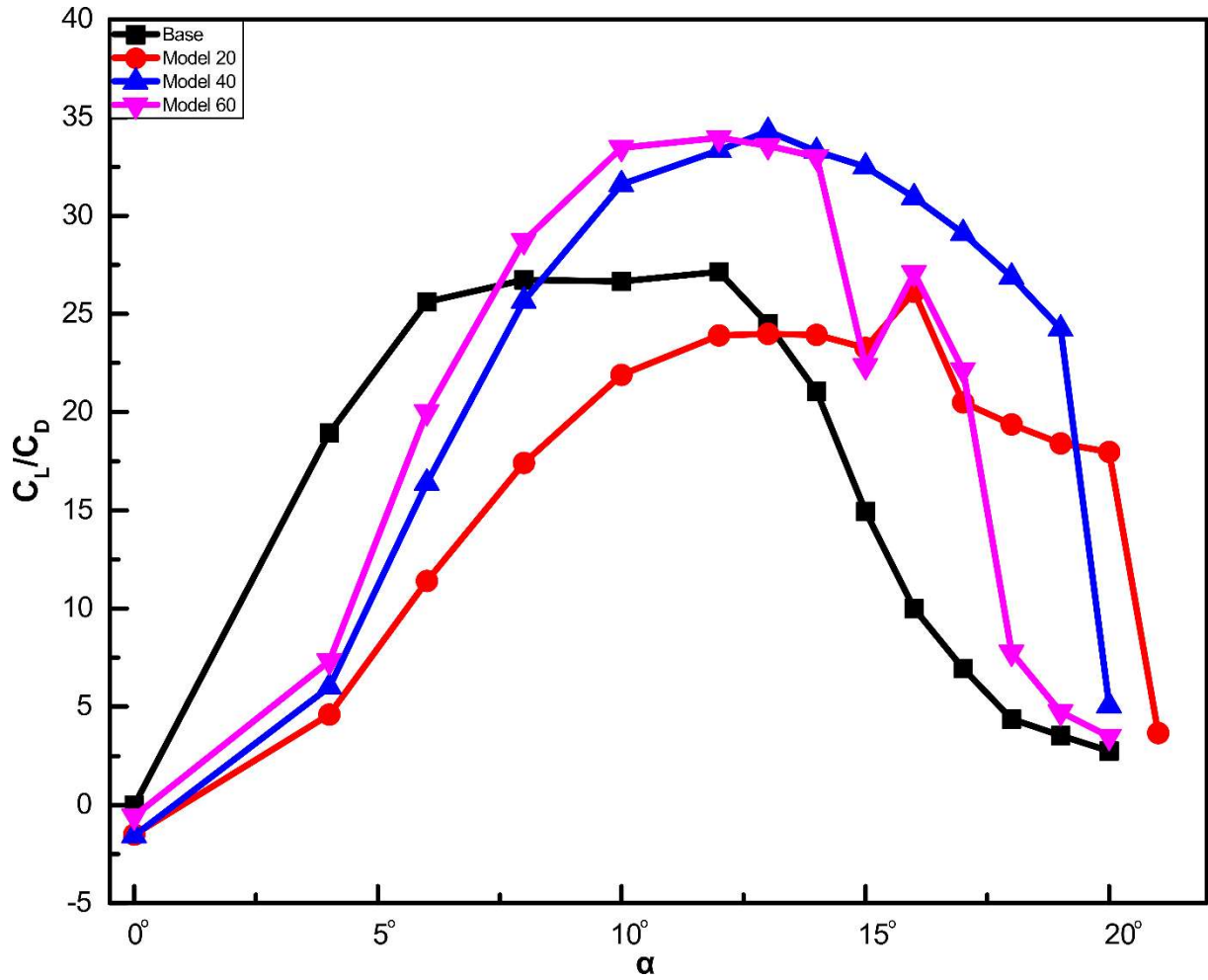


Figure 4.4. Variation of lift to drag ratio with the angles of Attack for different models

Figure 4.5a and 4.5b show pressure distributions on Model 20, Model 40 and Model 60 at $\alpha = 0^\circ, 4^\circ, 8^\circ, 10^\circ, 12^\circ, 15^\circ, 17^\circ$ and 19° in order to have a detailed understanding of flow dynamics such as trailing edge separation, transition and reattachment points. Obtained results are also compared with base airfoil in order to reveal effect of slot. It should be noted that the slotted airfoil consists of two parts, therefore pressure distributions are presented in two different curves. In the pressure diagram, the red lines represent the front part of the slotted airfoil, while the blue lines represent the rear part. Also, the black lines represent the base airfoil. It is clearly seen that the pressure distribution on the upper and lower side of base airfoil is same due to the symmetrical airfoil profile at $\alpha = 0^\circ$. Therefore lift coefficient is evaluated as zero. On the other hand, slot leads to higher pressure on the upper side of airfoil, so negative lift force occurs at $\alpha = 0^\circ$. At angle of attack of $\alpha = 4^\circ$, slot location remarkably affect the pressure distributions especially for front section of airfoil. It is seen that maximum $-C_p$ values increase with the

increasing slot location, however it is still lower than base airfoil. It is observed that lower side of front part enhances pressure difference with upper side. However, this situation leads to decrease in pressure for rear part since lower side of front part corresponding to upper side of rear part of airfoil. It is observed that the peak magnitude of pressure increase with the increasing angle of attack and pressure distribution on the upper side of front part gets similar trend with base airfoil. For Model 60, pressure distribution on the upper side of front part overlaps with the base airfoil at $\alpha = 8^\circ$. Moreover, the peak magnitude of pressure is higher than base airfoil. Therefore, C_L value is higher than base airfoil. At $\alpha = 10^\circ$, the pressure distributions of front part is similar with the base airfoil. It is observed that enhancing of lift force is dominated by rear part of airfoil for slotted models. Similar trend is also observed at $\alpha = 12^\circ$. At $\alpha = 15^\circ$, the peak magnitude of pressure increase with the increasing slot location. It is observed that the peak magnitude of pressure decreases for base airfoil when angle of attack increases from $\alpha = 15^\circ$ to 17° due to the stall condition. This situation is more distinct at 19° . It is clearly seen that slotted model remarkably delays the stall angle in comparison with the base airfoil. In controlled airfoil profiles, it was observed that the angle of attack $\alpha = 8^\circ$, where the pressure change is most evident according to the location of the slot's outlet. At this angle of attack, Model 20 and Model 40 cannot exceed the pressure coefficient of the base airfoil, while the Model 60 and base airfoil's C_p have overlapped. Unlike other angles of attack, at $\alpha = 10^\circ$, no change was observed at the top of the C_p graph depending on the outlet location of the slot. Also, another inconsistency in terms of the peak of the C_p graph is observed for Model 60 at the angle of attack of $\alpha = 19^\circ$. Although the pressure increased as the location of slot outlet was moved towards the trailing edge, it was observed that the pressure of the Model 60, whose slot's outlet is located more close to the trailing edge compared to Model 20 and 40, decreased.

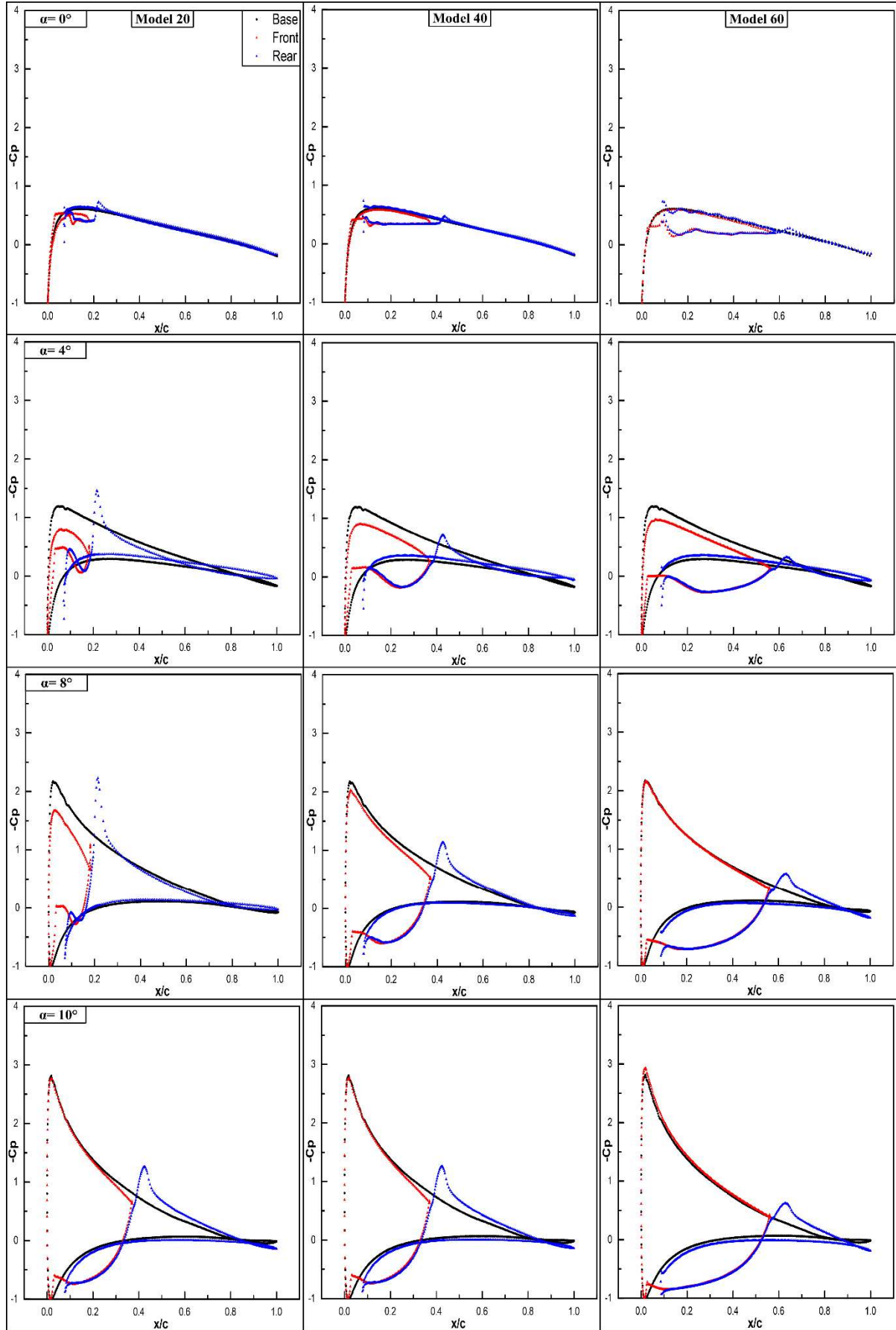


Figure 4.5a. Pressure Coefficient distributions for all Models at $0^\circ \leq \alpha \leq 10^\circ$ (k- ω SST)

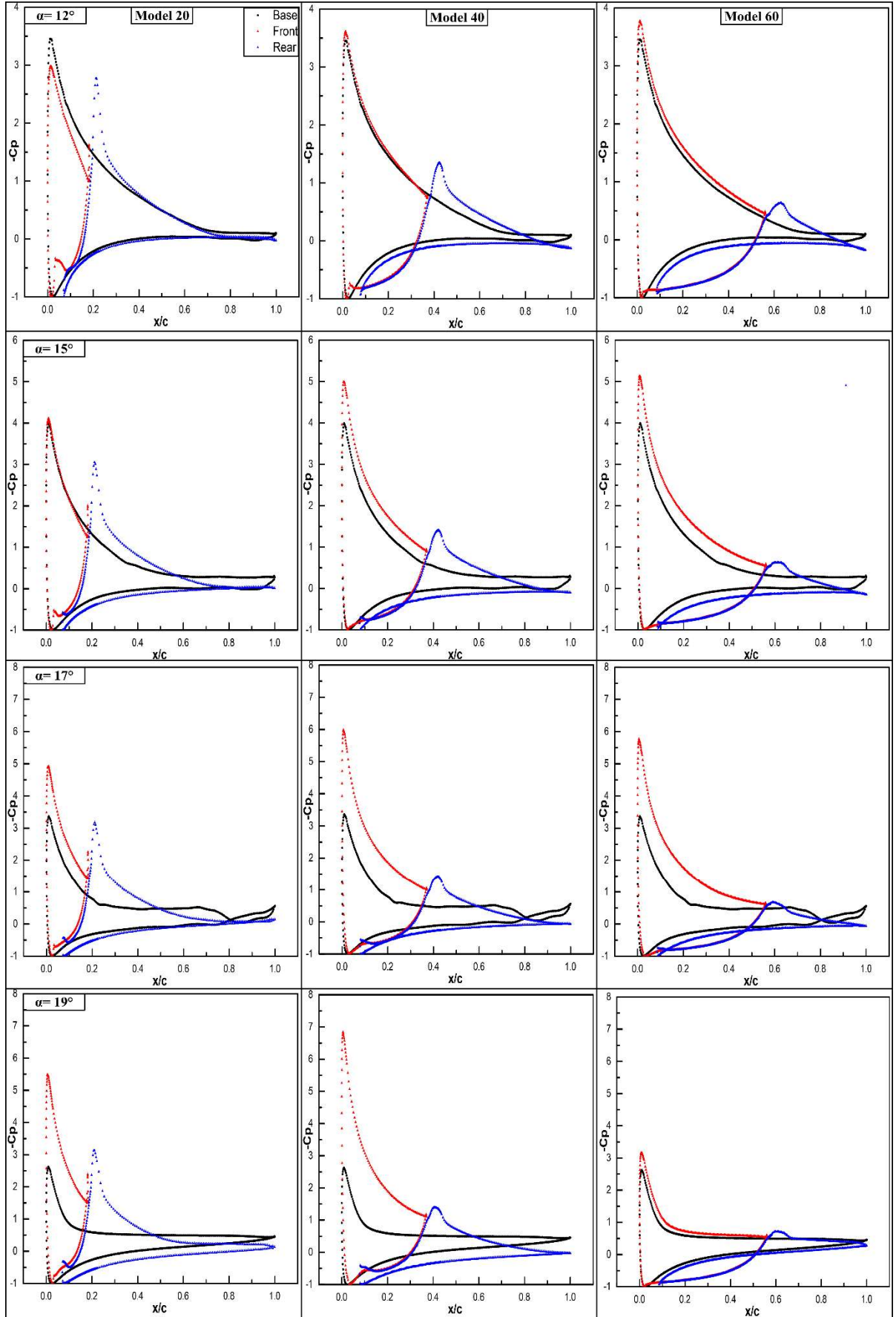


Figure 4.5b. Pressure Coefficient distributions for all Models at $12^\circ \leq \alpha \leq 19^\circ$ (k- ω SST)

In Figure 4.6, the streamline topologies obtained from $k-\omega$ SST turbulence model for all models are presented at angles of attack of $\alpha=4^\circ, 8^\circ, 10^\circ, 12^\circ, 15^\circ, 17^\circ$ and 19° . At $\alpha=4^\circ$, streamline patterns indicates that flow follows the airfoil surface for all models. For Model 40 and 60, small foci (which indicates the recirculation region) observed in the slot. A slight trailing edge separation appears for base airfoil at $\alpha=12^\circ$, while it is more distinct at $\alpha=15^\circ$. Furthermore, laminar separation bubble is not observed for base airfoil and this ones similar with the literature. For base airfoil, the separation point get closes the leading edge with the increasing angle of attack and recirculation region covers suction surface of airfoil. The trailing edge separation is also observed for Model 20 at $\alpha=15^\circ$.

On the other hand, foci formation on the trailing edge is not observed for Model 40 and 60 since transferring of momentum from slot prevent the trailing edge separation. The size of recirculation region increase at $\alpha=17^\circ$ and 19° , however, it is still smaller than base airfoil. For Model 60, two well defined foci is observed at $\alpha=19^\circ$. This flow pattern indicates that recirculation region aroused from stall is divided by slot. Trailing edge separation and also recirculation region on the section surface is not observed for Model 40 at all angles of attack. From this point of view, it can be concluded that model 40 is the most successful model for all angles of attack when compared to the other models. Trailing edge separation for the base airfoil at the angle of attack $\alpha=12^\circ$, could not be prevented by the Model 20. Model 40 and Model 60 were observed to be able to prevent this separation. A similar situation applies to the angles of attack of $\alpha=15^\circ$ and 17° ; however although the Model 20 could not eliminate this separation, it managed to significantly reduce it.

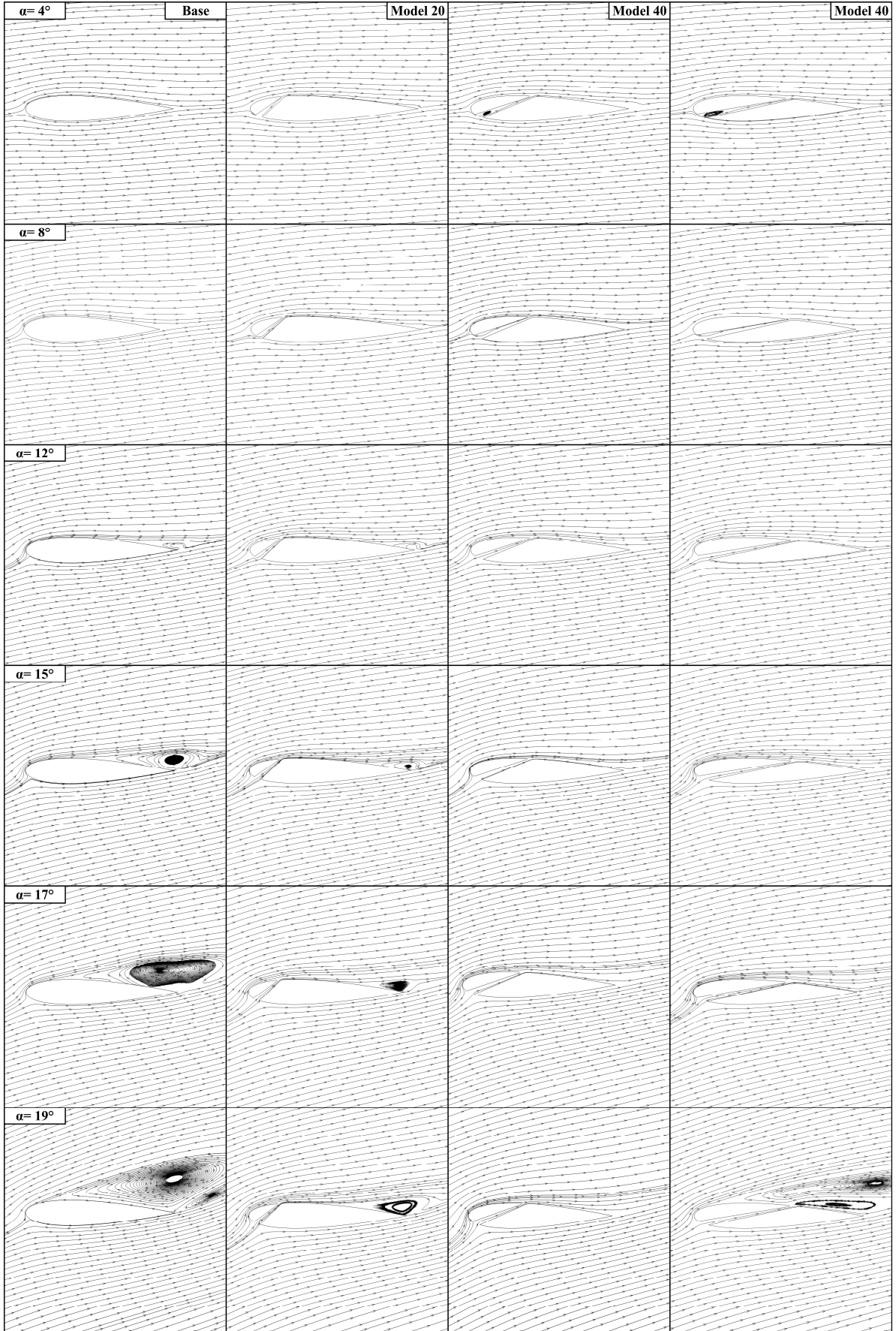


Figure 4.6. Streamline Topologies for different Models at variant angles of attack.

Figure 4.7 shows the stream-wise velocity contours for base airfoil and controlled airfoils (Model 20, Model 40 and Model 60) at the angles of attack of $\alpha=4^\circ, 8^\circ, 12^\circ, 15^\circ, 17^\circ$ and 19° . It is clearly seen that slot has insignificant effect on the flow structure around the airfoil for pre stall angles. At angle of attack of $\alpha=4^\circ$, flow around airfoil is disturbed by slot and size of low velocity region for slotted models is larger than base airfoil. Therefore, drag coefficients of slotted models are higher than base airfoil at $\alpha=4^\circ$. Moreover, low velocity region occurs at inlet section of slot since flow is blocked by slot at low angles of attack. At $\alpha=12^\circ$, low velocity region moves through the leading edge of base airfoil. However, flow emanating from slot enhances momentum transfer to suction surface and it hinders trailing edge separation for slotted models. Although low velocity region is suppressed by slot at $\alpha=15^\circ$, the point where low velocity region start is observed on the front part of airfoil for Model 60. It is also observed that the slot leads to increase in the flow velocity and it reach maximum value at the outlet of the slot due to the slot geometry. At $\alpha=17^\circ$ and $\alpha=19^\circ$, dark blue contour on the suction side of the base airfoil depicts recirculation region due to stall. For Model 20, this region smaller than the base airfoil. Although, recirculation region is not occur for Model 40, it covers suction surface of airfoil for Model 60 at $\alpha=19^\circ$ as observed from the base airfoil. It is observed that the angle of attack is $\alpha=15^\circ$, where the low velocity region is most successfully suppressed in common by all controlled airfoil models. If the models are evaluated among themselves, it is seen that the angle of attack of the Model 20 best suppresses the low velocity region is $\alpha=19^\circ$. Model 40 and 60 were observed to be $\alpha=17^\circ$ and $\alpha=12^\circ$, respectively. From these results, it is understood that the angle of attack at which the low velocity region is successfully suppressed is inversely proportional as the slot outlet approaches the trailing edge.

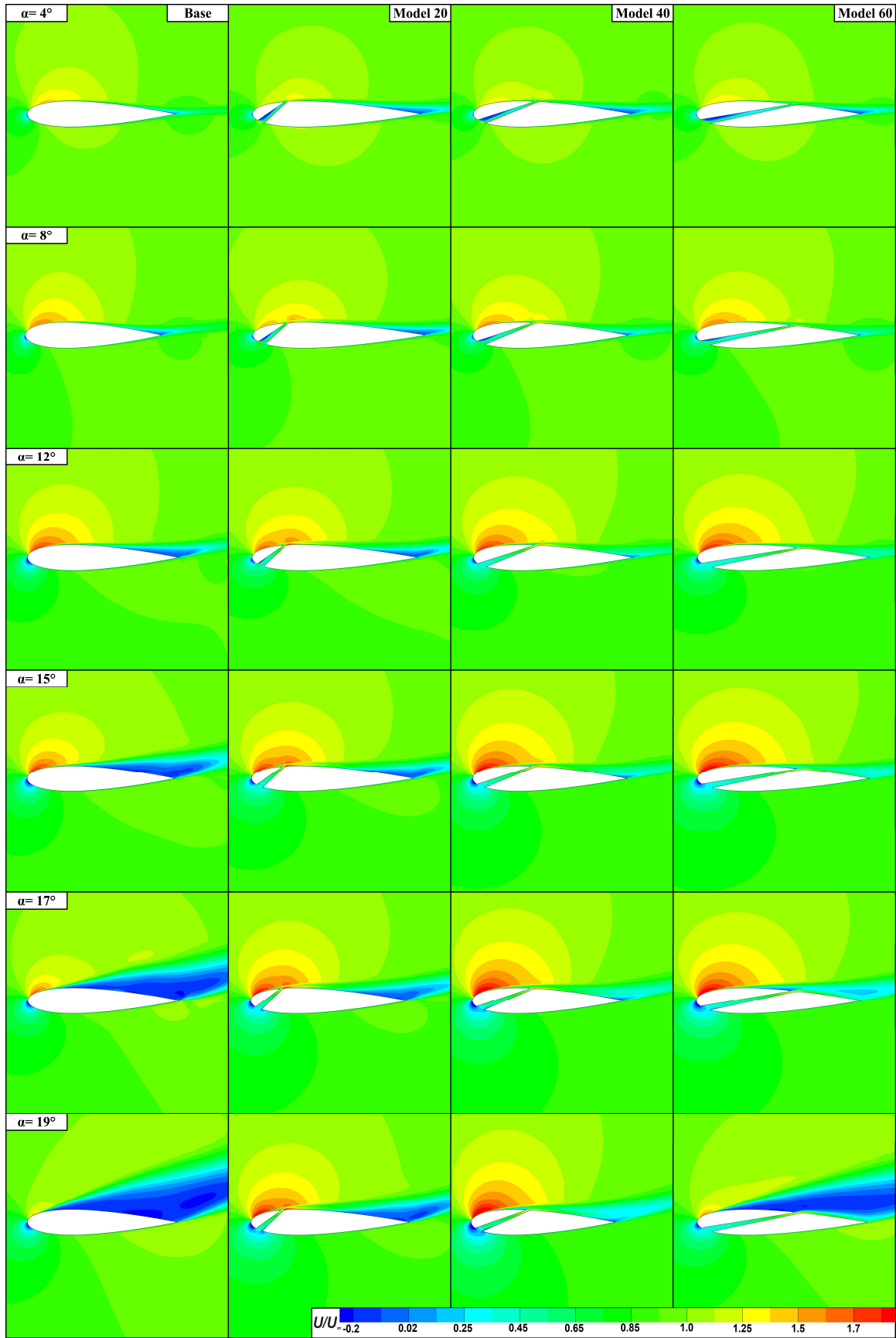


Figure 4.7. Stream-Wise Velocity Contours for Base, Model 20, Model 40 and Model 60 at $4^\circ \leq \alpha \leq 19^\circ$

The comparison of turbulence kinetic energy (TKE) contours for Base airfoil, Model 20, Model 40 and Model 60 at $\alpha = 0^\circ, 4^\circ, 8^\circ, 10^\circ$ are given in Figure 4.8a. Also, the TKE contours for all models at the angles of attack of $\alpha = 12^\circ, 15^\circ, 17^\circ, 19^\circ$ are shown in Figure 4.8b. The min, max and number of levels values in the range distribution section in Figure 4.8b have been changed to make the color distribution more meaningful. At $\alpha = 0^\circ$, TKE contours cluster in near wake region for base airfoil due to interaction of shear layers on the both side of airfoil. On the other hand, the peak magnitude of TKE occurs at inlet and outlet of slot since flow separation form at that locations. At $\alpha = 4^\circ$, slot adversely affects flow patterns and the peak magnitude of TKE remarkably increases in near wake region in comparison with base airfoil. The effect of slot on the suppression of TKE is observed at $\alpha = 8^\circ$ especially for Model 40 and 60. For high angles of attacks, the effect of slot on the suppression of TKE is more apparent when compared to the base airfoil. It is clearly seen that TKE is remarkably suppressed at $\alpha \geq 10^\circ$. It can be concluded, flow control around the airfoil can be achieved using of slot for high angles of attack.

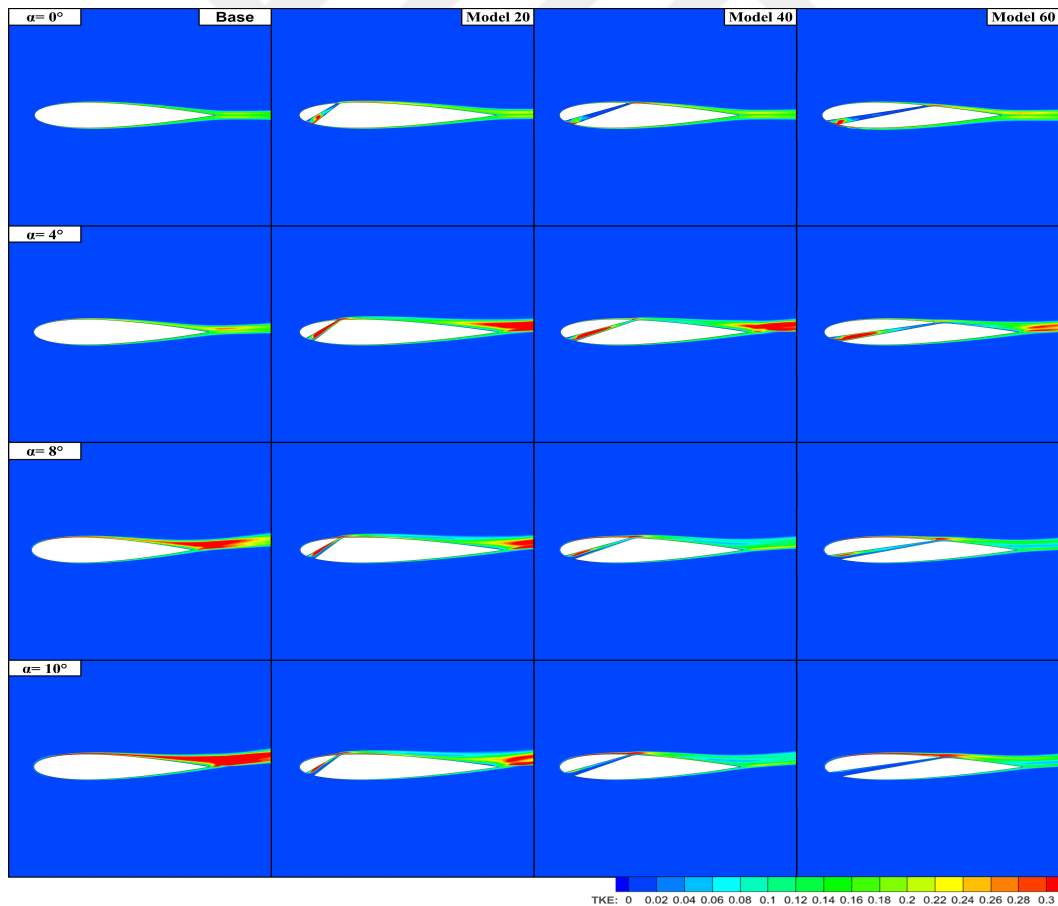


Figure 4.8a. Comparison of Turbulence Kinetic Energy contours for at the Angles of Attack of $0^\circ \leq \alpha \leq 10^\circ$

It was not observed that the base airfoil differs significantly from the controlled airfoils at the angles of attack of $\alpha = 0^\circ$ and 4° . While it is observed that turbulence intensity increases at the rear of the base airfoil at $\alpha = 8^\circ$, the increase in controlled models is much less. Especially after the angle of attack of $\alpha = 15^\circ$, the very intense redness behind the base airfoil shows how much turbulent kinetic energy is produced. While the turbulence intensities of Model 20 and Model 60 are similar to each other, turbulence intensity occurred in Model 40 is minimal.

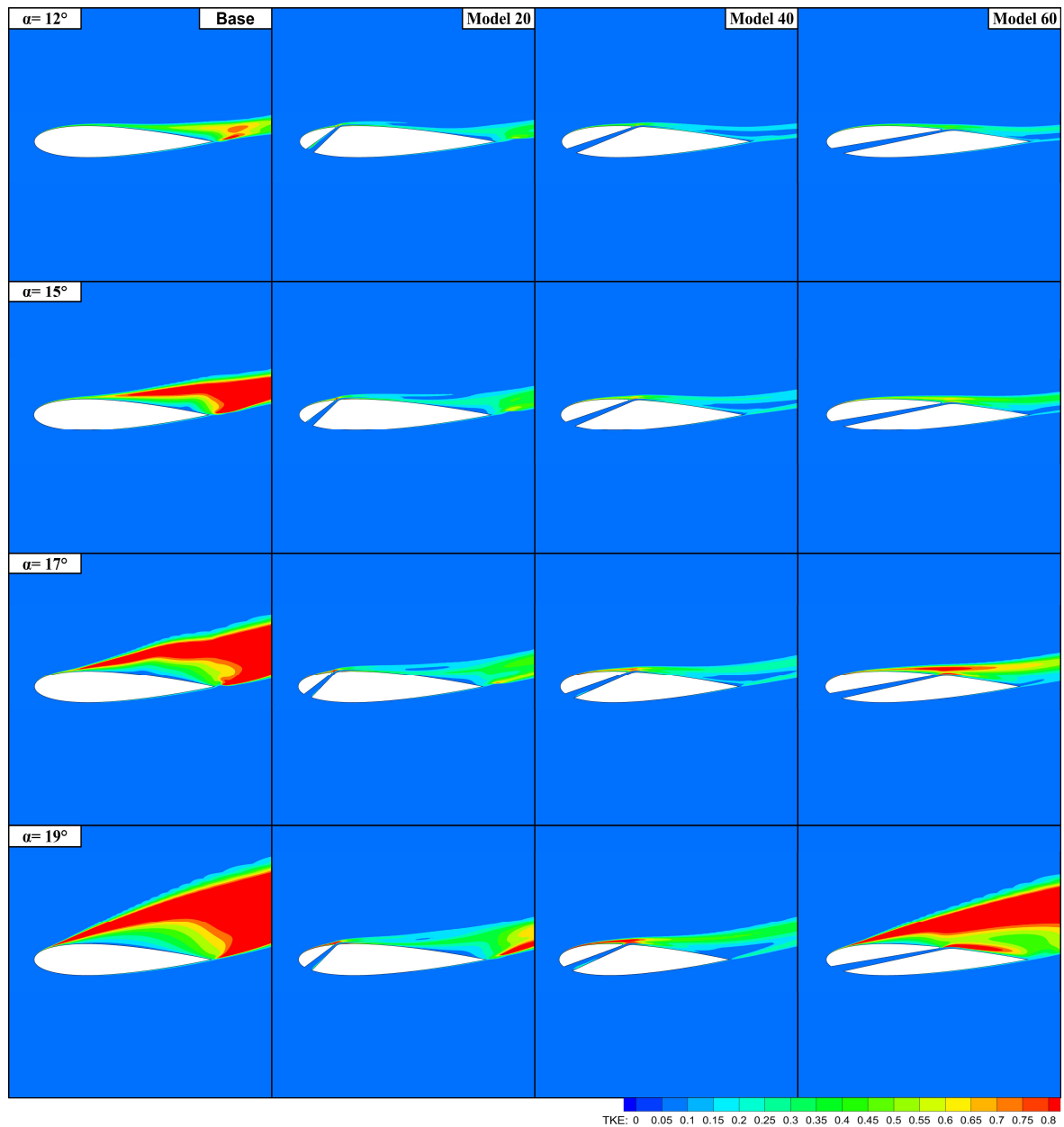


Figure 4.8b. Comparison of Turbulence Kinetic Energy contour at the Angles of Attack of $12^\circ \leq \alpha \leq 19^\circ$

4.2. K-ε RNG Model

The k-ε RNG turbulence model was chosen according to study of Mohamed et al. (2019) and Aziz et al. (2018). The results of the study of Mohamed et al. (2019) is compared with using k-ε RNG turbulence model for the angles of attack of $\alpha=10^\circ$ and $\alpha=15^\circ$. According to the comparison of the data supplied from the literature with results of the turbulence model, which is given by table 3.3, using k-ε RNG is become sensible.

Figure 4.9 compares the lift coefficients obtained from different models at variant the angles of attack. For the base airfoil, it is clearly seen that the lift coefficient is zero at the angle of attack of $\alpha=0^\circ$. The lift coefficient (C_L) increases as the angle of attack increases, up to $\alpha=19^\circ$. The highest lift coefficient is found at the angle of attack of $\alpha=18^\circ$ as $C_L = 1.3$ at. At the angles of attack of $\alpha=19^\circ$ and 20° the lift coefficient values are calculated as 1.28 and 1.18, respectively.

For controlled models, lift coefficients are below zero at the angle of attack of $\alpha=0^\circ$, since the airfoil aerodynamics are deteriorated due to the slot. However, it is clearly observed that higher lift coefficients are obtained especially after 15° as the angle of attack increase when compared to the base airfoil. In terms of lift coefficient highest value is obtained for Model 60 at angle of attacks range about 15° - 20° . It is seen that lift coefficient values of Model 20 are lower than other models, however, stall angle is higher than slotted models. Although Model 20 is the worst model in terms of lift coefficient value at low angles of attack it has a higher lift coefficient at higher angles of attack than other models. The lift coefficient of Model 20 increases continuously up to the angle of attack of $\alpha=25^\circ$, and then remains constant. Model 40 proves to be the most successful model for angles of attack between $\alpha=20^\circ$ and $\alpha=24^\circ$. It is seen that all the controlled models have a higher lift coefficient than the base airfoil at high angles of attack.

The comparison of drag coefficients at different angle of attack is given in Figure 4.10. The base airfoil appears to have a lower drag coefficient at low angles of attack compared to the controlled airfoil models. With the increase in the angle of attack, this situation is reversed. It is seen that the drag coefficient for the base airfoil is inversely proportional to the angle of attack and increases dramatically, especially after $\alpha=15^\circ$. In controlled models, Model 20 has a higher drag coefficient compared to other models at low angles of attack, while, it has a lower coefficient at high angles of attack. Moreover, the drag coefficient of Model 40 increases by an average of 4% between $\alpha=0^\circ$ and 17° , and increased by 6% at $\alpha=17^\circ$. Up to angle of attack of

$\alpha=17^\circ$, the value of the drag coefficient were fluctuated. At the angle of attack of $\alpha=0^\circ$ highest drag coefficient ($C_D=0.035$) is evaluated for Model 60 while it has a lower drag coefficients at low angles of attack in comparison with other models. The highest coefficient of drag is calculated as 0.12 at $\alpha=21^\circ$.

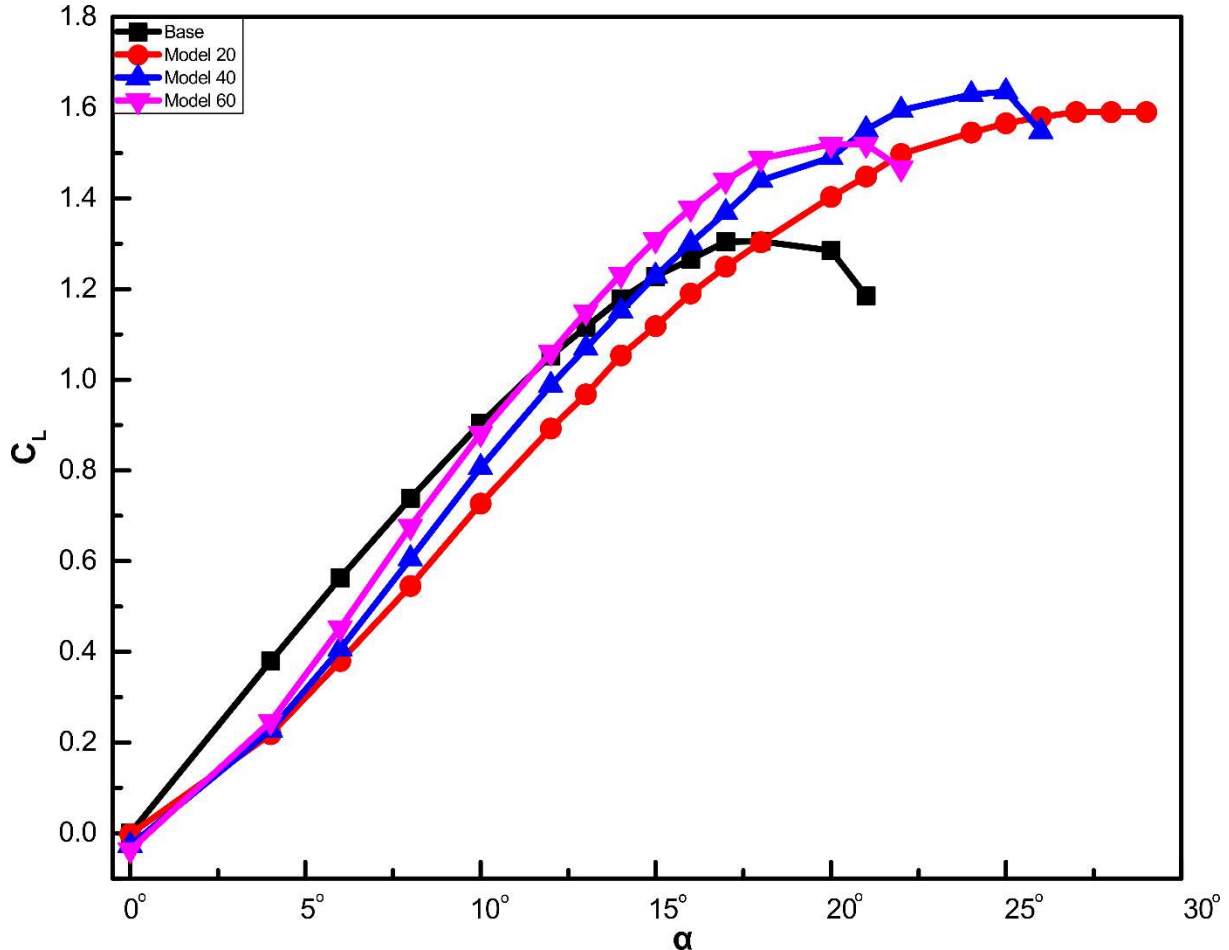


Figure 4.9. Variation of lift Coefficient with the angles of Attack for different models

Figure 4.11 demonstrates the ratio of the lift to drag coefficient at various the angle of attack for all models. It is clearly seen that the ratio of the lift coefficient to the drag coefficient of the base airfoil is higher at angles of attack between $\alpha=0^\circ$ and 13° compared to the controlled airfoils. For base airfoil, this ratio starts to decrease after the angles of attack of $\alpha=13^\circ$. Although this ratio decrease after 15° for controlled models, their values are higher than the base airfoil at high angles of attack. It should also be noted that the maximum value of C_L/C_D obtained from the base airfoil ($C_L/C_D=23.4$) at the angle of attack of $\alpha=10^\circ$ is not procured by the slotted models at all angles. The highest ratio is obtained for Model 60 at $\alpha=13^\circ$ when compared to

the slotted cases. For Model 20 and 40, the highest ratio is evaluated at 14° and 15° , respectively.

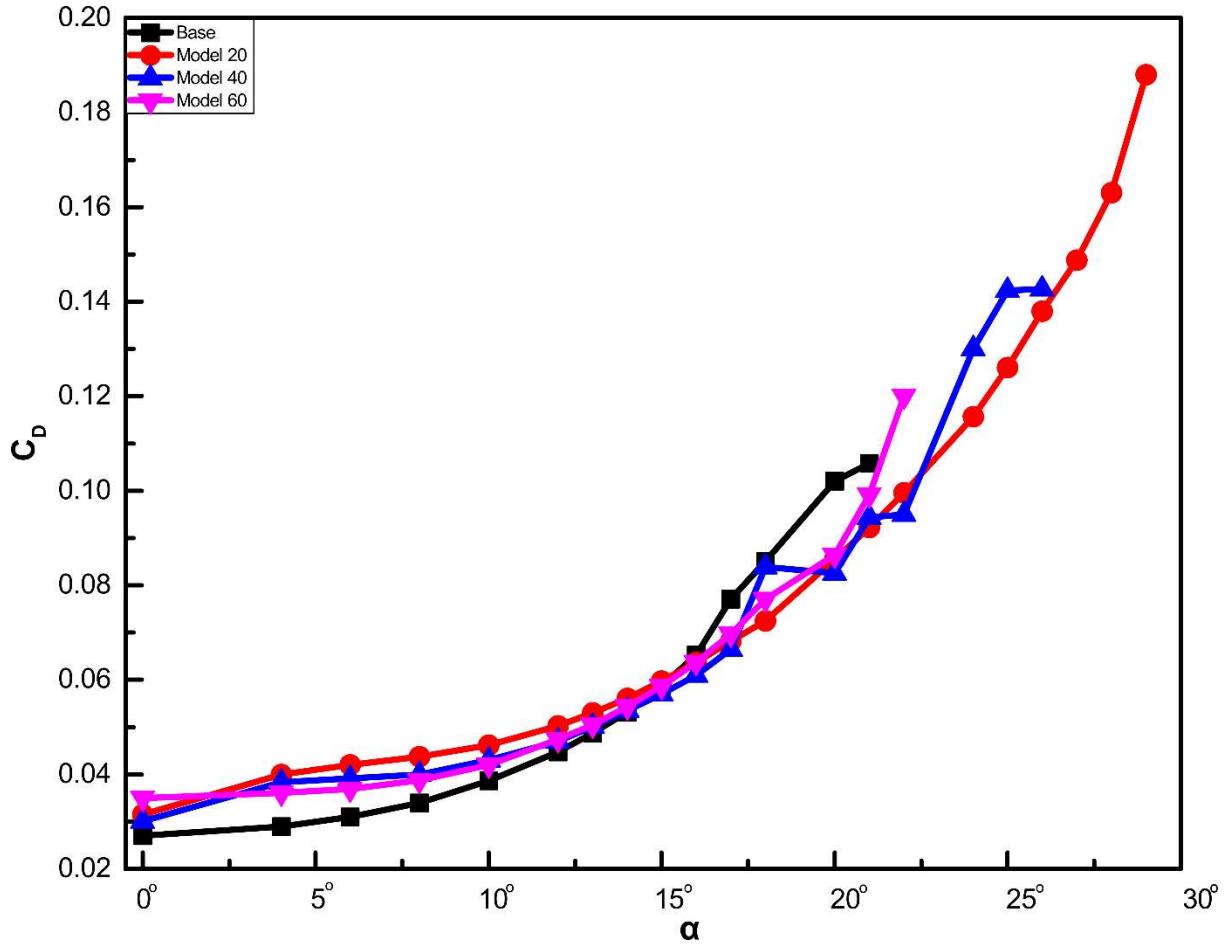


Figure 4.10. Variation of drag Coefficient with the angles of Attack for different models

Figure 4.12a and 4.12b show pressure distributions on Model 20, Model 40 and Model 60 at $\alpha = 0^\circ, 4^\circ, 8^\circ, 10^\circ, 12^\circ, 15^\circ, 17^\circ, 19^\circ$. These pressure distribution graphs are important data for understanding fluid dynamics. In the Figures, each column represent different models while each row depicts different angles of attack. The controlled airfoils were divided into two parts, therefore pressure distributions were presented separately. In figures, the red line represents the front part of the slotted airfoil, while the blue lines represent the rear part. Also, the pressure distribution of base airfoil is drawn by black lines.

As mentioned in the first chapter, due to the symmetrical geometry of NACA 0018 at $\alpha = 0^\circ$, the pressure distribution on the suction and pressure surfaces is the same. Therefore, the lift coefficient is evaluated as zero.

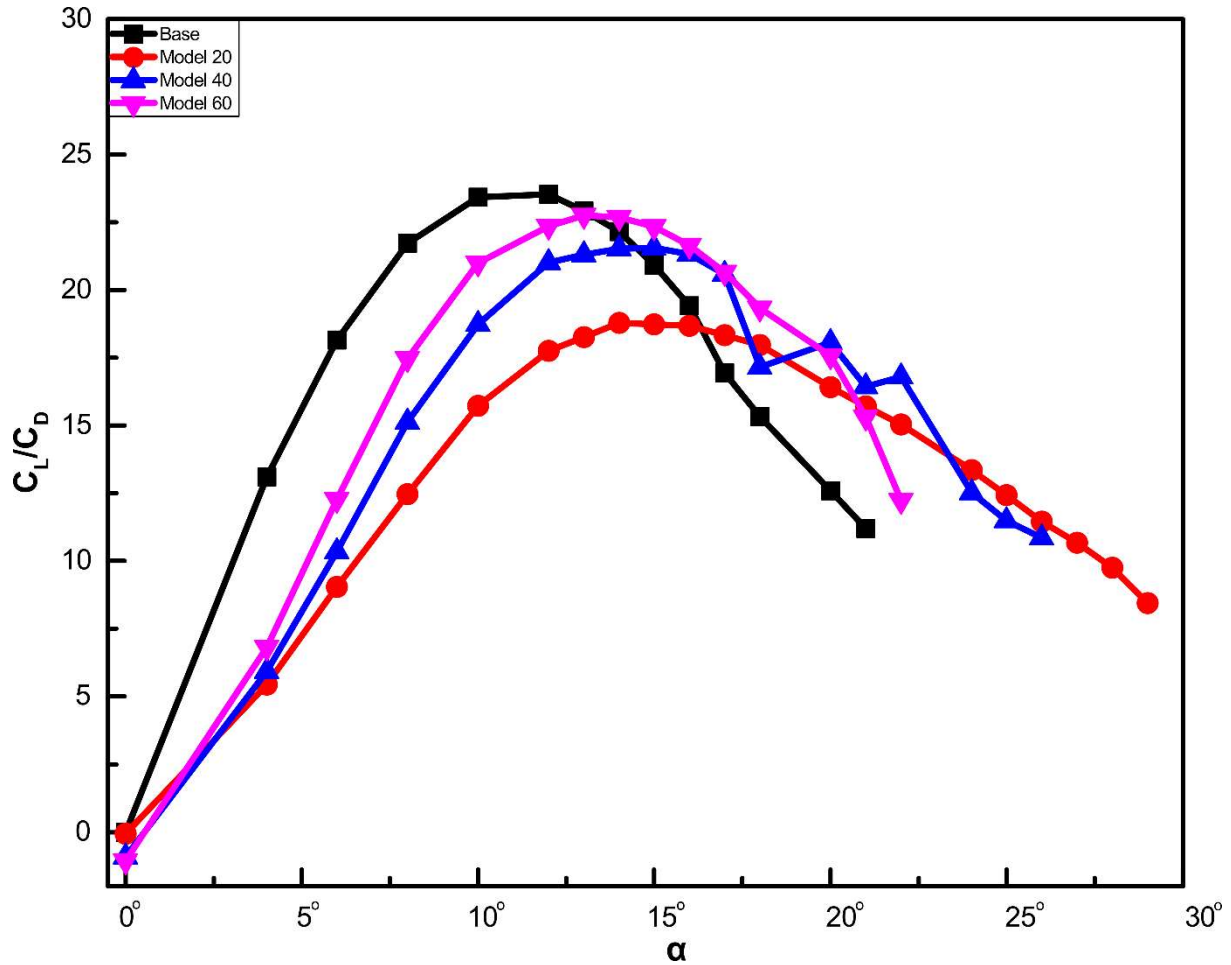


Figure 4.11. Variation of lift Coefficient over drag Coefficient with the angles of Attack for different models

The results obtained from the $k-\epsilon$ RNG model indicates that the pressure distributions, especially for the front part of the airfoil, are significantly affected by the slot position at all angles of attack as obtained from the $k-\omega$ SST model. It is also seen that the maximum negative pressure coefficient values of front part increase as the the location of slot outlet approaches the trailing edge. The pressure distribution profiles obtained from the $k-\epsilon$ RNG model are nearly similar with that obtained from $k-\omega$ SST model up to $\alpha = 8^\circ$. For Model 20, the magnitudes of pressure coefficient of front part are lower than the baser airfoil at $\alpha = 10^\circ$ in contrary to $k-\omega$ SST model. However, the peak magnitude of $-C_p$ of rear part is higher than obtained from $k-\omega$ SST model. For Model 40 and 60, the pressure distribution on the suction side of front part is nearly same with the base airfoil at $\geq 12^\circ$. It is observed that flow is dominated by rear part for these cases.

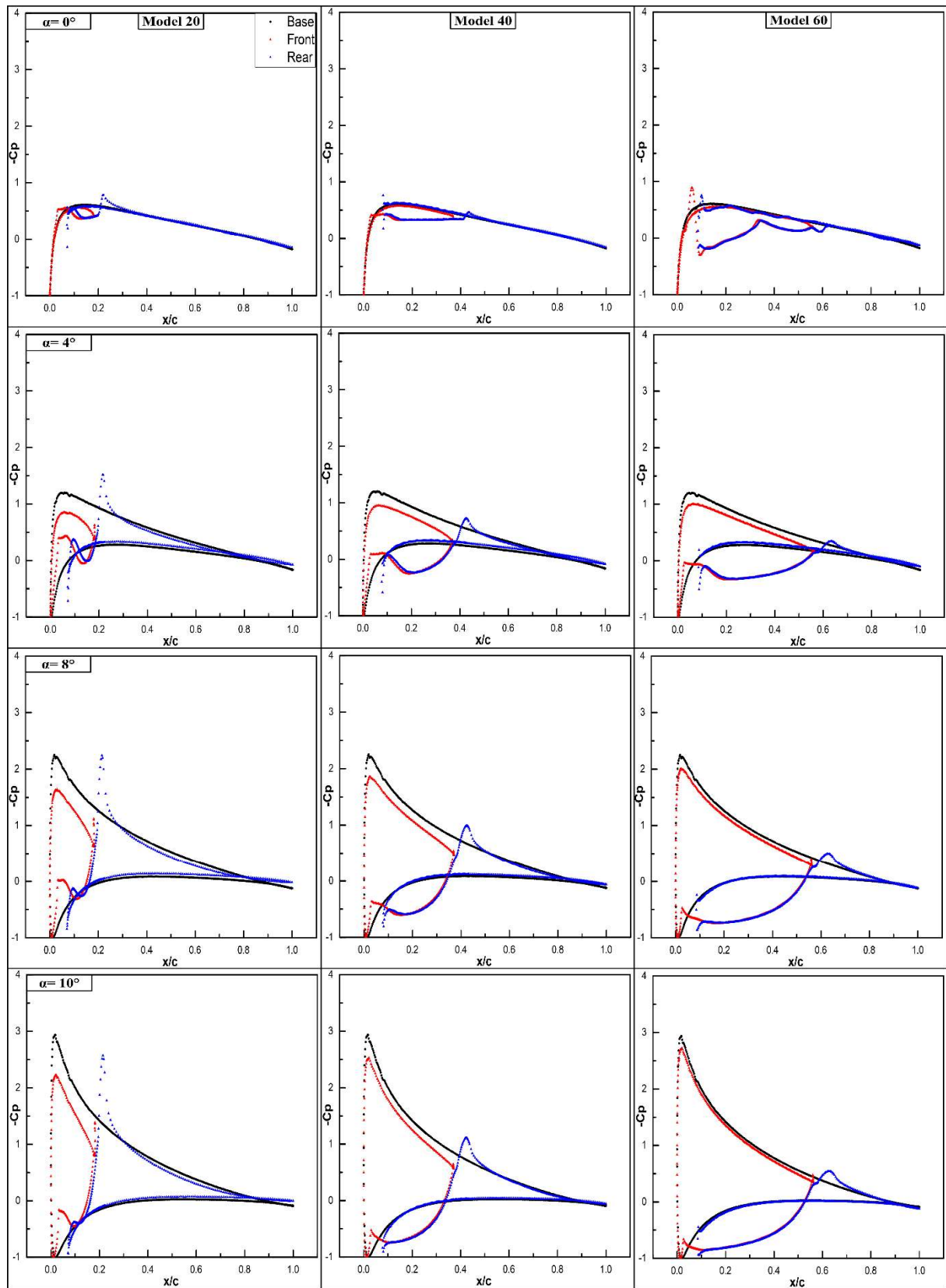


Figure 4.12a. Pressure Coefficient distributions for all Models at $0^\circ \leq \alpha \leq 10^\circ$ (k- ϵ RNG)

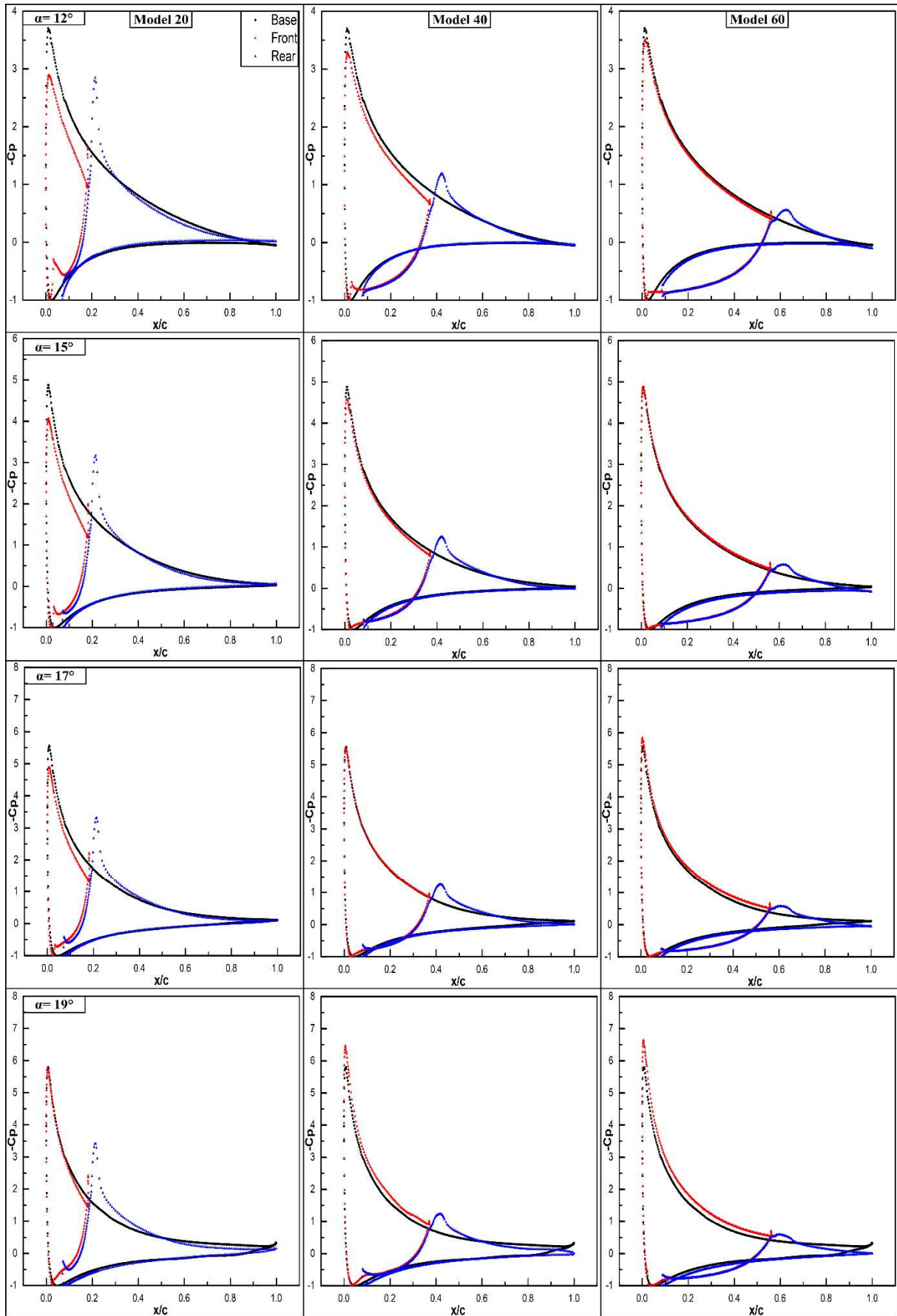


Figure 4.12b. Pressure Coefficient distributions for all Models at $12^\circ \leq \alpha \leq 19^\circ$ (k- ϵ RNG)

In Figure 4.13, the streamline topologies obtained from k- ϵ RNG turbulence model for base and three controlled airfoil models (Model 20, Model 40 and Model 60) are presented at angles of attack of $\alpha=4^\circ$, 8° , 10° , 12° , 15° , 17° and 19° . For this turbulence model (k- ϵ RNG), foci formation on the trailing edge is not observed up to $\alpha=17^\circ$ while it appears at $\alpha=15^\circ$ for k- ω SST model. Furthermore, the size of recirculation region obtained from k- ϵ RNG turbulence model is smaller than that obtained from k- ω SST model at $\alpha=17^\circ$ and 19° . For Model 40 and 60, small recirculation region is observed in the slot. For Model 20, trailing edge separation is emerged at the angle of attack of $\alpha=15^\circ$. The the size of this recirculation region increases with increasing angle of attack and reaches its maximum size at $\alpha=19^\circ$. However, the size of recirculation region of Model 20 is relatively smaller than the base airfoil at $\alpha=19^\circ$. On the other hand, foci formation is not observed for Model 40 and 60. Unlike the K- ω SST turbulence model, the K- ϵ RNG turbulence model prevents trailing edge separation in all angles of attack for Models 40 and 60. The streamline topology of Model 20 is similar to the topology obtained from the K- ω SST model.

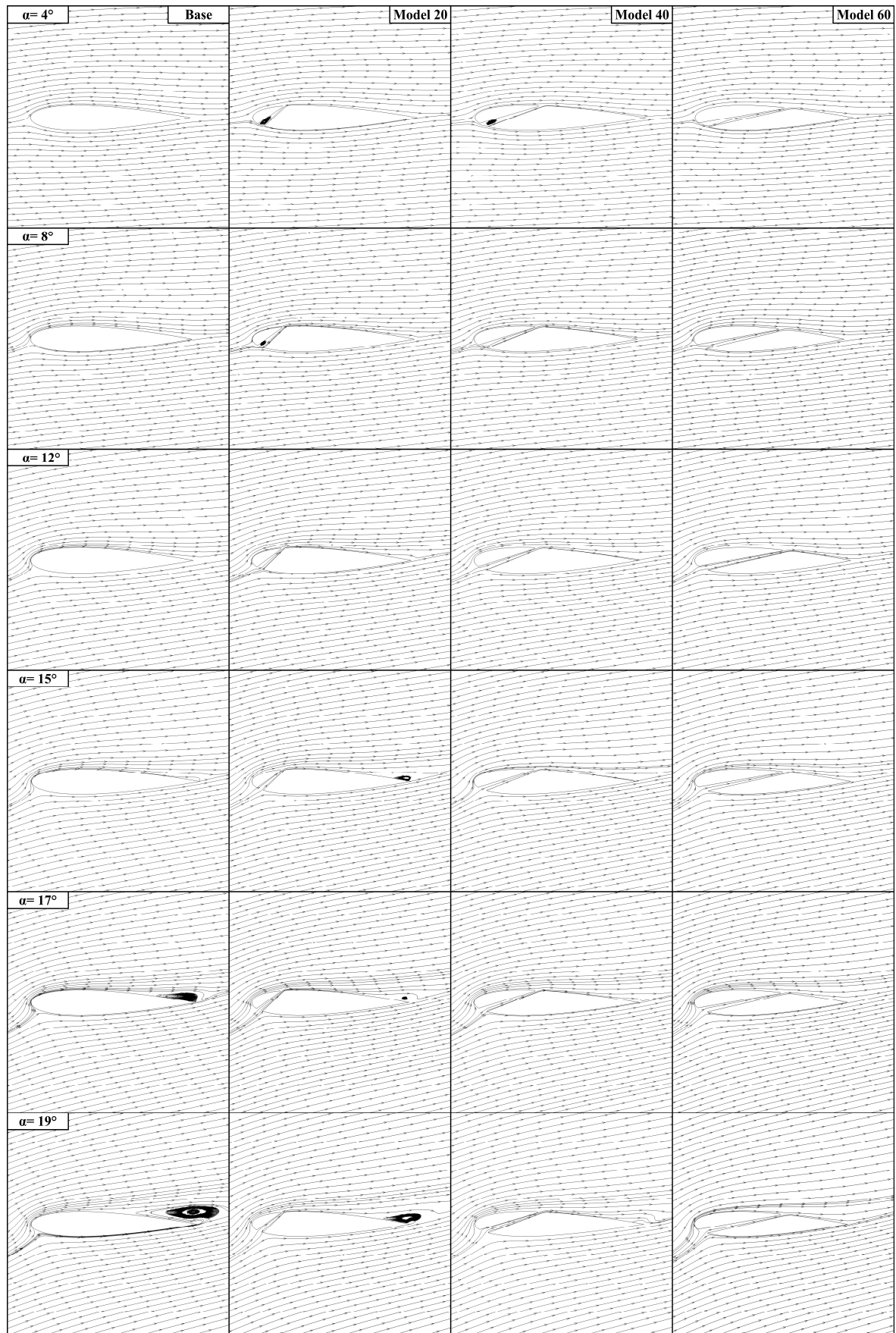


Figure 4.13. Streamline Topologies for different Models at variant angles of attack.

Figure 4.14 shows the Stream-wise velocity contours for base airfoil and controlled airfoils (Model 20, Model 40 and Model 60) at the angles of attack of $\alpha=4^\circ, 8^\circ, 12^\circ, 15^\circ, 17^\circ$ and 19° . At low angles of attack, it is observed that the slot does not have a positive effect on the wing aerodynamics. The flow separation occurs at inlet section of slot and does not gain momentum inside the slot, therefore it approaches the stopping point at low angles of attack. This leads to an increase in the drag coefficient when compared to the base airfoil. Momentum transfer from the slot enhances with increasing angle of attack since the inlet section of slot coincides with the flow direction. At $\alpha=17^\circ$ and $\alpha=19^\circ$, the region seen as blue in the contour distribution due to the formation of the recirculation region on the suction side of the Model 40 is smaller than the base airfoil due to the momentum transfer. According to the results obtained from the k- ϵ RNG turbulence model, Model 40 is the most successful model and this good agreement with the k- ω SST. However, flow structure over the airfoil is different for higher angle of attacks. If the models are evaluated among themselves, it is seen that the Model 20 cannot suppress the low velocity region up to the $\alpha=19^\circ$ angle of attack and has a contour distribution close to the base airfoil. While the Model 40 is able to suppress the low velocity region at every angle of attack. Also, the Model 60 is observed to be more effective at low angles of attack than at high angles of attack. From these results, it is seen that the model that can suppress the low velocity region most successfully is the Model 40.

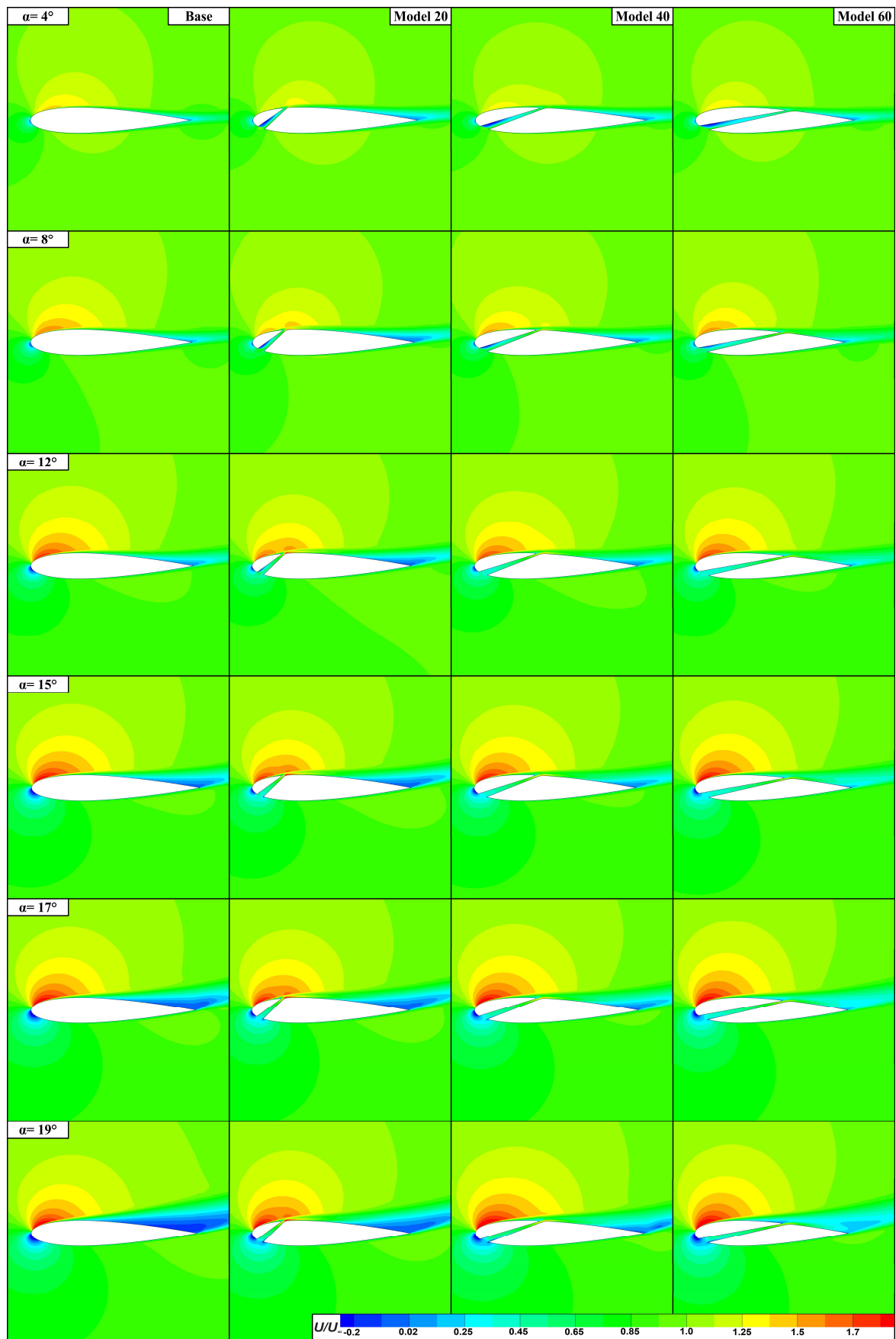


Figure 4.14. Stream-Wise Velocity Contours for all models at $4^\circ \leq \alpha \leq 19^\circ$

Comparison of Turbulence Kinetic Energy (TKE) contours for Base airfoil, Model 20, Model 40 and Model 60 at $\alpha = 0^\circ, 4^\circ, 8^\circ, 10^\circ$ are given in Figure 4.15a. Also, the TKE contours for all models at the angles of attack of $\alpha = 12^\circ, 15^\circ, 17^\circ, 19^\circ$ are shown in Figure 4.15b. It should be noted that the min, max and number of levels values in the range distribution section in Figure 4.15b was changed to make the color distribution more meaningful. At low angles of attack for base airfoil, TKE contours concentrates in near wake region due to interaction of shear layers on pressure and suction side of airfoil. High concentrates contours are observed at the slot inlet and outlet sections. It is also seen from the TKE contour that the slot negatively affects the flow structure for all models. It is observed that as the angle of attack increases, the peak magnitude of TKE in the wake region of base the airfoil increases, while it decreases for slotted model up to $\alpha = 8^\circ$. A similar situation is also observed in the solution of the other turbulence model, the $k-\omega$ sst model. However, TKE contours having high magnitude is more apparent on the front part at $\alpha = 8^\circ$ is seen for Model 40 and 60.

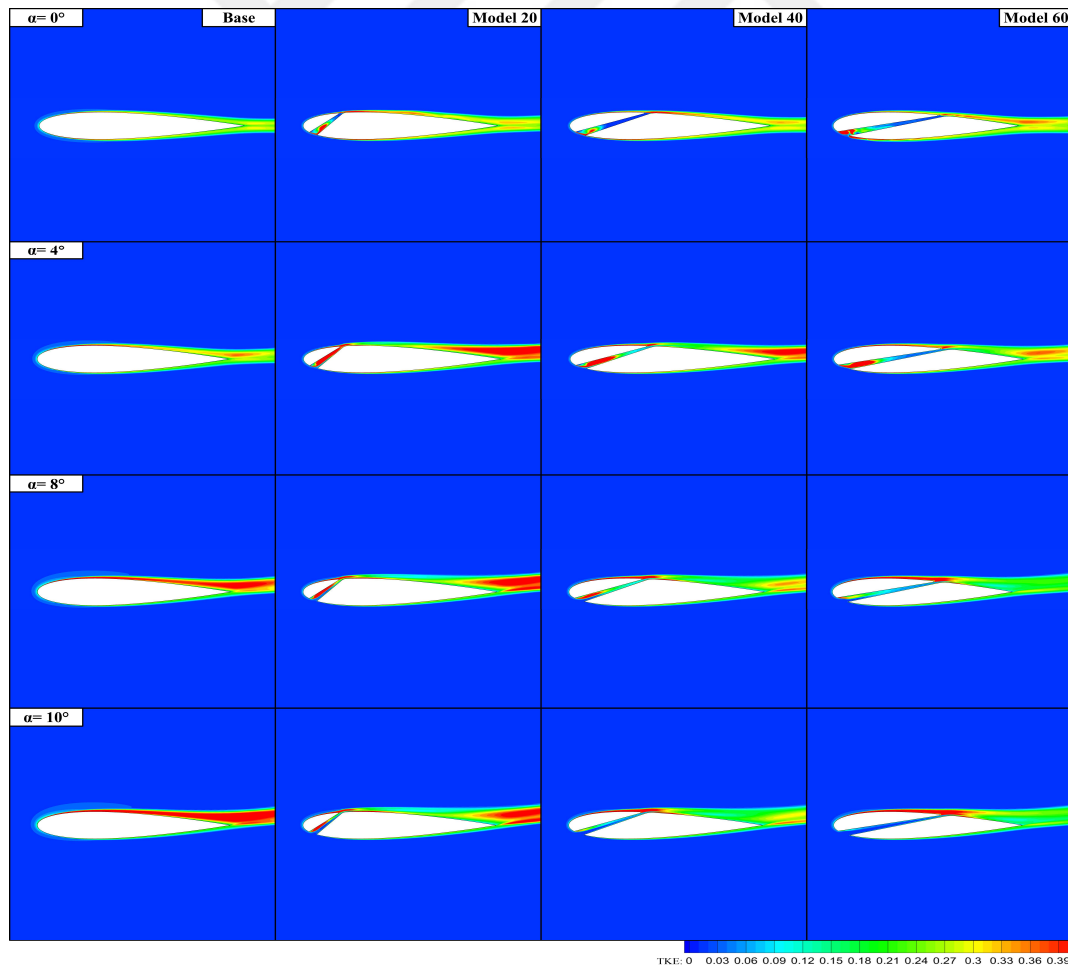


Figure 4.15a. Comparison of Turbulence Kinetic Energy at the Angles of Attack of $0^\circ \leq \alpha \leq 10^\circ$

It has been observed that as the angle of attack increases, the peak magnitude of turbulent kinetic energy contours increases and high concentrate TKE contours cover suction surface of airfoil. When controlled airfoils are examined, it is seen that the peak magnitude of turbulent kinetic energy locates on the front part of airfoil especially for high angles of attack. This situation indicates that slot has insignificant effect on the leading edge separation. However, TKE is remarkably suppressed on the wake region of airfoil when compared to the base airfoil. The results obtained from the k- ϵ RNG model indicate that the slotted models are more successful at the post-stall angles of attack.

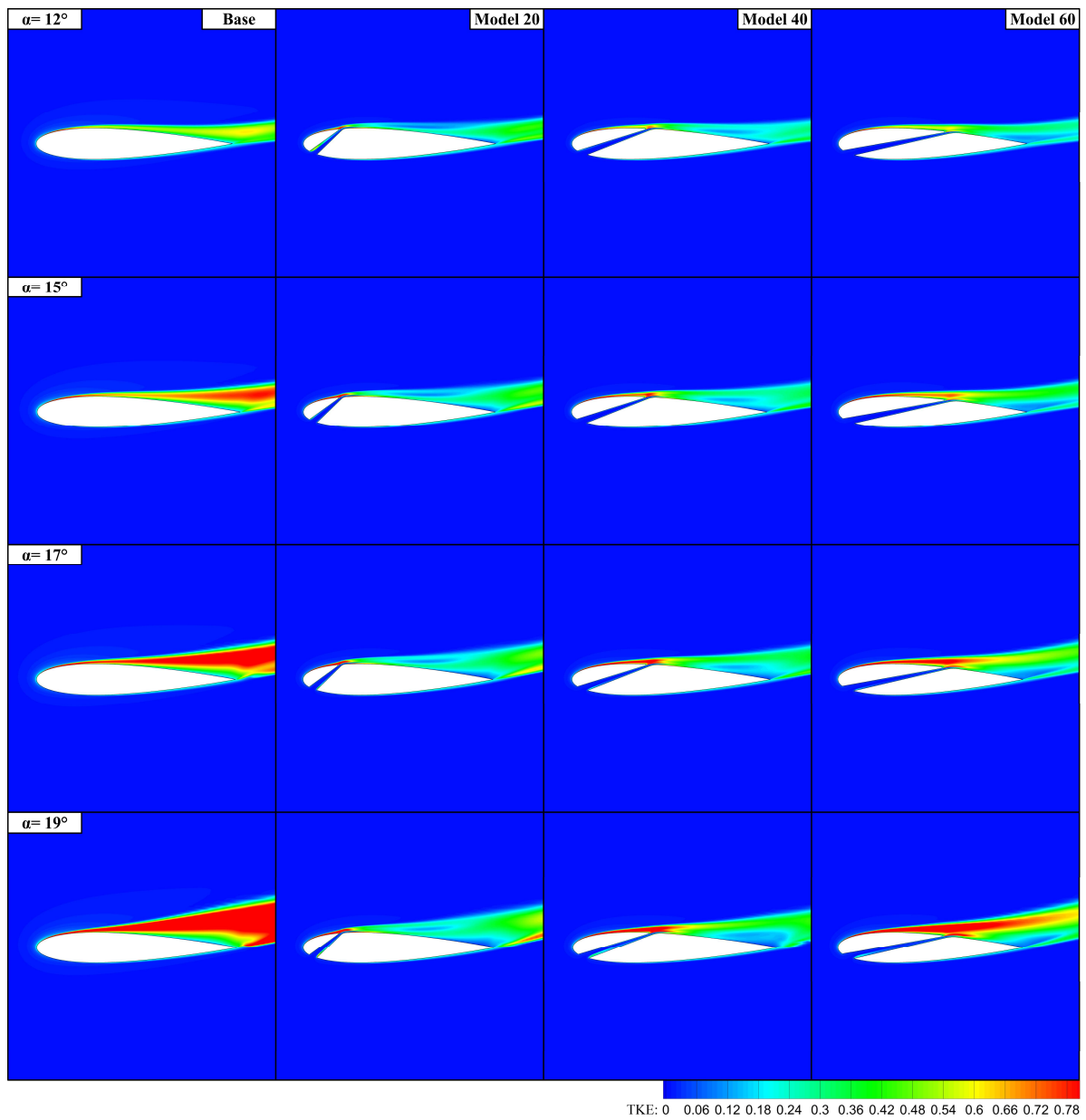


Figure 4.15b. Comparison of Turbulence Kinetic Energy at the Angles of Attack of $12^\circ \leq \alpha \leq 19^\circ$

4.3. Comparison of K- ω SST and K- ϵ RNG Turbulence Models

In order to better reveal the effect of turbulence models, the results are given comparatively in this section. In this numerical study, the closest result to the experimental results of Timmer (2008) with the results obtained from the K- ω SST turbulence model for base airfoil. The biggest error in the K- ϵ RNG turbulence model is that the base airfoil does not stall at the angle of attack of $\alpha = 16^\circ$, which is the stall angle. Although the drag coefficient showed an abnormal increase in this angle of attack, it is not observed from the streamline topologies and pressure coefficient distributions. Furthermore, the table 3.3 containing the results of a study with two different angles of attack of $\alpha = 10^\circ$ and $\alpha = 15^\circ$ in the literature for the slot-controlled NACA 0018 airfoil profile is shown in the previous section. As seen in this table, the results obtained from the K- ϵ RNG turbulence model overlap with the results in the literature, showing that the results of this model are reliable for slotted airfoil profiles.

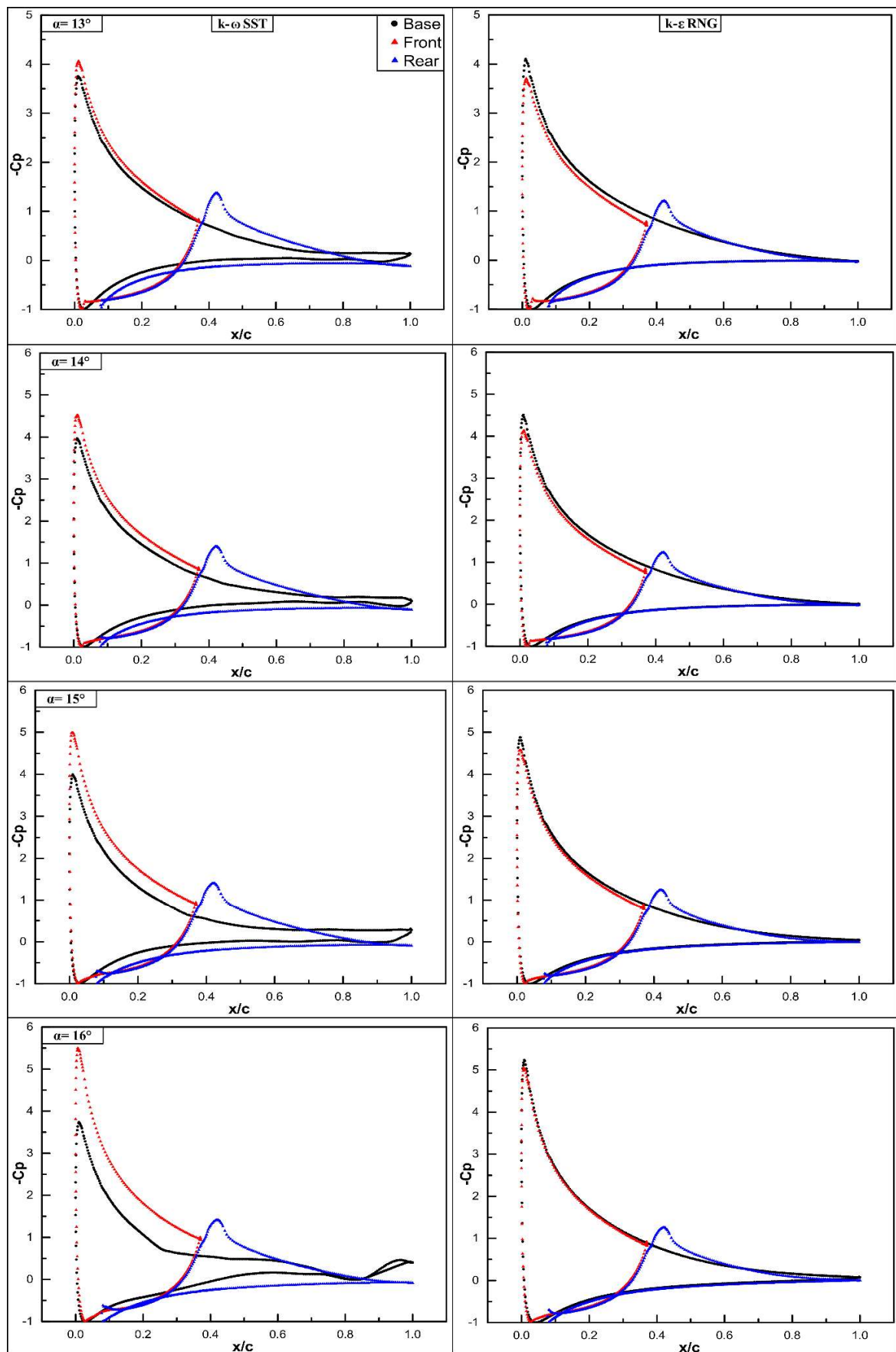


Figure 4.16. Comparison of Coefficient of Pressure (C_p) for Base and Model 40.

Figure 4.16 shows the pressure distribution graphs obtained from numerical solutions using K- ω SST and K- ε RNG turbulence model side by side for Model 40's cases at the angles of attack of $\alpha=13^\circ, 14^\circ, 15^\circ, 16^\circ$. The left column of the graph shows the the results obtained from K- ω SST turbulence, while the right column depicts the results of K- ε RNG turbulence model. At $\alpha=13^\circ, 14^\circ$, the maximum value obtained from K- ε RNG is higher than K- ω SST for base airfoil on the contrary to Model 40. For K- ω SST turbulent model, all the angles of attack the pressure distributions are higher than base airfoil; however, for K- ε RNG turbulent model, the pressure distribution on suction side of front part of Model 40 is almost the same with the base airfoil. Also, the maximum value obtained from K- ω SST is higher than K- ε RNG for Model 40 at $\alpha \geq 15^\circ$.

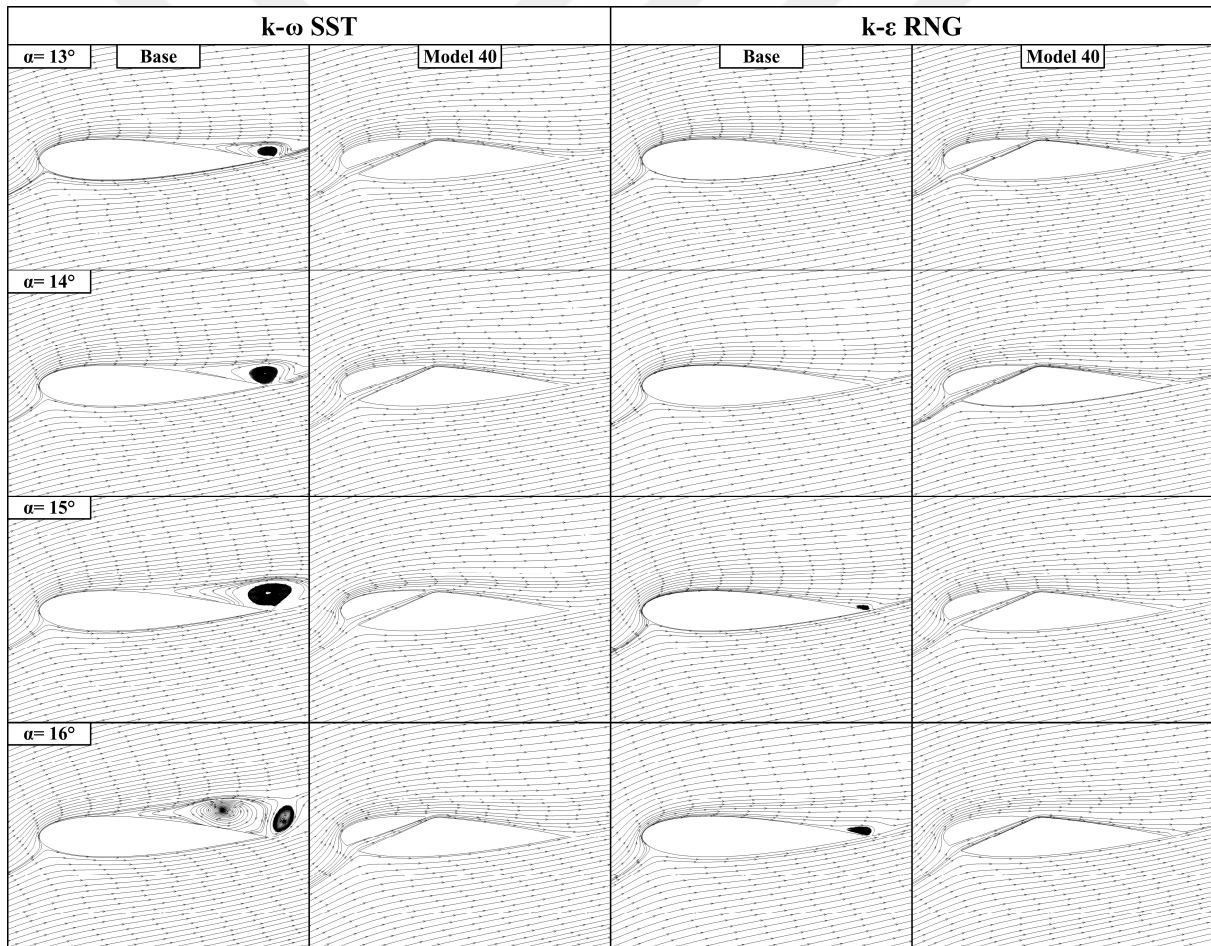


Figure 4.17. Streamlines topologies obtained from K- ω SST and K- ε RNG turbulence models for $13^\circ \leq \alpha \leq 16^\circ$

Figure 4.17 presents the streamline topologies obtained from two different turbulence models, for the base airfoil and the Model 40 model at the angles of attack of $\alpha = 13^\circ, 14^\circ, 15^\circ$ and 16° .

For the base airfoil, K- ω SST turbulence model seems to capture trailing edge separation at 13° , whereas the K- ε RNG model seems to capture it only at 15° and 16° . In terms of K- ω SST turbulence model, it is observed that the two foci formation occurs at angle of attack of $\alpha = 16^\circ$. For K- ε RNG model, however, this recirculation formation occurred relatively small and the two foci formations are not observed. For Model 40, foci formation on the trailing edge is not captured by K- ω SST turbulence model. The examination in terms of Model 40, the results obtained with the K- ω SST turbulence model depicts that TKE is remarkably suppressed by slot. Also, a similar results is obtained with the K- ε RNG model.

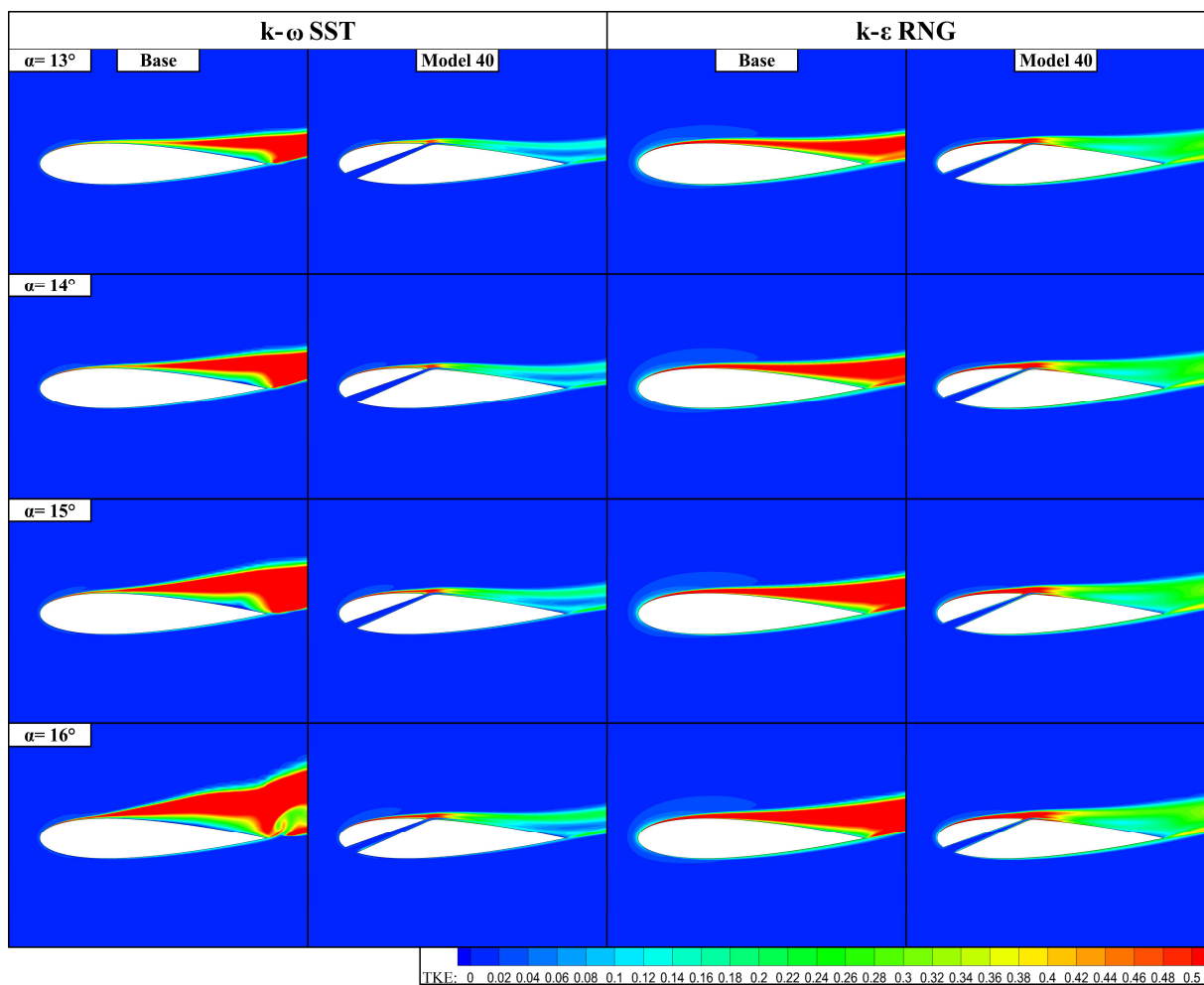


Figure 4.18. Comparison of Turbulence Kinetic Energy (TKE)

The comparison of Turbulent Kinetic energy contours obtained from different turbulence models for base and Model at the angles of attack of $\alpha = 13^\circ, 14^\circ, 15^\circ$ and 16° are given in Figure 4.180. For base airfoil the main difference is obtained for the size of high TKE clusters. The results of K- ω SST turbulence model depicts that high TKE region is larger than K- ε RNG for

base airfoil. The examination in terms of Model 40, the results obtained with the $K-\omega$ SST turbulence model depicts that TKE is remarkably suppressed by slot. Also, a similar results is obtained with the $K-\epsilon$ RNG model. On the other hand, the size of high TKE contours located on the front part is larger than $K-\omega$ SST.



5. CONCLUSIONS

In this study, it was aimed to suppress the recirculation area formed over the airfoil at high angles of attack using passive flow control technique. In this content, the effects of slot on flow over NACA 0018 airfoil profile was numerically investigated by using ANSYS FLUENT software. The parameters were determined as the outlet location of the slot (L1) and the angle of attack (α). Also, two different turbulence models were examined which were K- ω SST and K- ϵ RNG. Reynolds number was set to $Re = 3 \times 10^5$. First of all, to determine the inlet location of the slot, the base airfoil profile was analyzed at various angles of attack. Saddle point was approximately obtained at $x/c = 0.05$ for high angle of attacks and inlet of slot was posinated that location for all models. The size of inlet section of the slot was determined to be four times of outlet section. The inlet location was kept constant for each models, and the outlet location on the suction side was designed to be $x/c = 20\%$ for Model 20, $x/c = 40\%$ for Model 40 and $x/c = 60\%$ for Model 60.

Although the controlled airfoils has in significant effect on the aerodynamic coefficients at low angles of attack in comparison with the base airfoils, it was observed that lift coefficient remarkably increases at the post-stall angles of attack. It was found that recirculation region over the airfoil was eliminated for Model 40 since transferring of momentum from slot prevent the trailing edge separation. According to the results of the K- ω SST turbulence model of Model 40 with slot outlet location $x/c = 0.4$, the stall angle was delayed up to 20° when compared to the base airfoil. Also, it was observed that the lift coefficient was increased by 8%, 40% and 154% at $\alpha = 10^\circ$, 15° and 19° respectively. A similar situation was observed in the K- ϵ RNG turbulence model. It was found that raitos of C_L/C_D of Model 60 are higher than other models especially at angle of attack of $\alpha > 15^\circ$ angles of attack. On the other hand, it was observed that as the angle of attack increases, the peak magnitude of turbulent kinetic energy (TKE) in the wake region of base the airfoil increases, while it decreases for slotted models up to angle of attack of $\alpha = 8^\circ$.

According to the results obtained from both K- ω SST and K- ϵ RNG turbulence models, it can be concluded that the slot location is important parameter to control flow around airfoil. Also the effect of slot on the lift coefficient incresase at higher angle of attack. Stall angle is delayed

for all models. Most effective location is determined as $x/c=40\%$ in order to suppress recirculation region and increase lift coefficient.



6. RECOMMENDATIONS

In the light of the results obtained from the presented thesis, the following suggestions can be made for future studies regarding the flow around the airfoils at low Re number flows:

- Flow characteristics on the suction side of the airfoil can be experimentally examined in detail.
- Studies can be performed on different airfoil profiles, Reynolds numbers, slot inlet/outlet ratio or location of the slot.
- Three dimensional unsteady numerical studies can be performed to reveal complex flow structure around slotted airfoil.
- The performance of slotted airfoil can be evaluated for wind turbine.

REFERENCES

- Abbasi, S., & Souri, M. (2020). Reducing aerodynamic noise in a rod-airfoil using suction and blowing control method. *International Journal of Applied Mechanics*, 12(04), 2050036.
- Acarer, S. (2020). Peak lift-to-drag ratio enhancement of the DU12W262 airfoil by passive flow control and its impact on horizontal and vertical axis wind turbines. *Energy*, 201, 117659.
- Açikel, H. H., & Genc, M. S. (2018). Control of laminar separation bubble over wind turbine airfoil using partial flexibility on suction surface. *Energy*, 165, 176-190.
- Açıkkar, M., & Sivrikaya, O. (2018). Artificial neural networks for estimation of the gross calorific value of Turkish lignite coal. In *3rd International Mediterranean Science and Engineering Congress, Adana, Turkey* (pp. 1075-1079).
- Airplane Aerodynamics, 2021. <http://www.free-online-private-pilot-ground-school.com/aerodynamics.html>
- Aley, K. S., Guha, T. K., & Kumar, R. (2020). Active flow control of a high-lift supercritical airfoil with microjet actuators. *AIAA Journal*, 58(5), 2053-2069.
- An, F. P., Balantekin, A. B., Band, H. R., Beriguete, W., Bishai, M., Blyth, S., ... & Rosero, R. (2014). Spectral measurement of electron antineutrino oscillation amplitude and frequency at Daya Bay. *Physical review letters*, 112(6), 061801.
- Andan, A. D., & Lee, D. J. (2018). Effect of external acoustic excitation on NACA0015 discrete tonal noise. *Applied Acoustics*, 141, 374-381.
- Atzori, M., Vinuesa, R., Fahland, G., Stroh, A., Gatti, D., Frohnäpfel, B., & Schlatter, P. (2020). Aerodynamic effects of uniform blowing and suction on a NACA4412 airfoil. *Flow, Turbulence and Combustion*, 105(3), 735-759.
- Auld D.J., Srinivas K., Aerodynamics For Students, 2008, Webtextbook.

Bangalore, A., & Sankar, L. N. (1996). Numerical analysis of aerodynamic performance of rotors with leading edge slats. *Computational mechanics*, 17(5), 335-342.

Bangalore, A., & Sankar, L. N. (1997). Forward-flight analysis of slatted rotors using Navier-Stokes methods. *Journal of aircraft*, 34(1), 80-86.

Baragona, M., Boermans, L. M. M., van Tooren, M. J. L., Bijl, H., & Beukers, A. (2003). Bubble bursting and stall hysteresis on single-slotted flap high-lift configuration. *AIAA journal*, 41(7), 1230-1237.

Bechert, D. W., Bruse, M., Hage, W., & Meyer, R. (2000). Fluid mechanics of biological surfaces and their technological application. *Naturwissenschaften*, 87(4), 157-171.

Belamadi, R., Djemili, A., Ilinca, A., & Mdouki, R. (2016). Aerodynamic performance analysis of slotted airfoils for application to wind turbine blades. *Journal of wind engineering and industrial aerodynamics*, 151, 79-99.

Biedermann, T. M., Chong, T. P., Kameier, F., & Paschereit, C. O. (2017). Statistical–empirical modeling of airfoil noise subjected to leading-edge serrations. *AIAA Journal*, 55(9), 3128-3142.

Bragg, M. B., & Gregorek, G. M. (1987). Experimental study of airfoil performance with vortex generators. *Journal of aircraft*, 24(5), 305-309.

Catalano, P., de Rosa, D., Mele, B., Tognaccini, R., & Moens, F. (2020). Performance improvements of a regional aircraft by riblets and natural laminar flow. *Journal of Aircraft*, 57(1), 29-40.

Cattafesta, L., Alvi, F., Williams, D., & Rowley, C. (2003). Review of active control of flow-induced cavity oscillations. In *33rd AIAA Fluid Dynamics Conference and Exhibit* (p. 3567).

Cleaver, D. J., Gursul, I., Calderon, D. E., & Wang, Z. (2014). Thrust enhancement due to flexible trailing-edge of plunging foils. *Journal of Fluids and Structures*, 51, 401-412.

Combes, T. P., Malik, A. S., Bramesfeld, G., & McQuilling, M. W. (2015). Efficient fluid-structure interaction method for conceptual design of flexible, fixed-wing micro-air-vehicle wings. *AIAA Journal*, 53(6), 1442-1454.

Daniel, L., & Traub, L. W. (2013). Effect of aspect ratio on gurney-flap performance. *Journal of aircraft*, 50(4), 1217-1225.

Dean, B., & Bhushan, B. (2010). Shark-skin surfaces for fluid-drag reduction in turbulent flow: a review. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 368(1929), 4775-4806.

Fatahian, E., Lohrasbi Nichkoohi, A., Salarian, H., & Khaleghinia, J. (2019). Comparative study of flow separation control using suction and blowing over an airfoil with/without flap. *Sādhanā*, 44(11), 1-19.

Fatehi, M., Nili-Ahmadabadi, M., Nematollahi, O., Minaiean, A., & Kim, K. C. (2019). Aerodynamic performance improvement of wind turbine blade by cavity shape optimization. *Renewable Energy*, 132, 773-785.

Fawzi, S., & Ali, A. H. Slot Design Procedure for a NACA 4412 at High Angle of Attack.

Genç, M. S., Demir, H., Özden, M., & Bodur, T. M. (2021). Experimental analysis of fluid-structure interaction in flexible wings at low Reynolds number flows. *Aircraft Engineering and Aerospace Technology*.

Hu, J., Wang, R., & Huang, D. (2018). Flow control mechanisms of a combined approach using blade slot and vortex generator in compressor cascade. *Aerospace Science and Technology*, 78, 320-331.

ITO, S. (2009). Aerodynamic influence of leading-edge serrations on an airfoil in a low Reynolds number-A study of an owl wing with leading edge serrations. *Journal of Biomechanical Science and Engineering*, 4(1), 117-123.

Jahanmiri, M. (2010). Active flow control: a review.

Jang, C. S., Ross, J. C., & Cummings, R. M. (1998). Numerical investigation of an airfoil with a Gurney flap. *Aircraft Design*, 1(2), 75-88.

Jiaqiang, E., Luo, B., Han, D., Chen, J., Liao, G., Zhang, F., & Ding, J. (2022). A comprehensive review on performance improvement of micro energy mechanical system: Heat transfer, micro combustion and energy conversion. *Energy*, 239, 122509.

Katz, J., Plotkin, A., 1991, Low-Speed Aerodynamics from Wing Theory to Panel Methods, McGraw-Hill, Inc, 632 pp.

Kerho, M., Hutcherson, S., Blackwelder, R. F., & Liebeck, R. H. (1993). Vortex generators used to control laminar separation bubbles. *Journal of aircraft*, 30(3), 315-319.

Kral, L. D. (2000). Active flow control technology. ASME Fluids Engineering Technical Brief.

Kutty, H. A., Rajendran, P., & Mule, A. (2017, April). Performance analysis of small scale UAV propeller with slotted design. In *2017 2nd International Conference for Convergence in Technology (I2CT)* (pp. 695-700). IEEE.

Lam, G. C., & Leung, R. C. (2018). Aeroacoustics of NACA 0018 airfoil with a cavity. *AIAA Journal*, 56(12), 4775-4786.

LeBeau, R., Karam, A., Pern, N., & Jacob, J. (2010). Analysis of low speed flow over an adaptive airfoil with oscillating camber. In *48th AIAA aerospace sciences meeting including the new horizons forum and aerospace exposition* (p. 93).

Lee, S. J., & Jang, Y. G. (2005). Control of flow around a NACA 0012 airfoil with a micro-riblet film. *Journal of fluids and structures*, 20(5), 659-672.

Li, D., Li, S., Xue, Y., Yang, Y., Su, W., Xia, Z., ... & Duan, H. (2014). The effect of slip distribution on flow past a circular cylinder. *Journal of Fluids and Structures*, 51, 211-224.

Li, X. L., Tang, G. H., Fan, Y. H., Yang, D. L., & Wang, S. Q. (2019). Numerical analysis of slotted airfoil fins for printed circuit heat exchanger in S-CO₂ Brayton cycle. *Journal of Nuclear Engineering and Radiation Science*, 5(4).

Li, Y. C., Wang, J. J., & Hua, J. (2007). Experimental investigations on the effects of divergent trailing edge and Gurney flaps on a supercritical airfoil. *Aerospace Science and Technology*, 11(2-3), 91-99.

Li, Z., Yao, Y., Wang, Q., Cai, J., Wu, Y., & Wang, H. (2019). Effect of internal helical-rib roughness on mixed convection flow and heat transfer in heated horizontal pipe flow of supercritical water. *International Journal of Heat and Mass Transfer*, 130, 1272-1287.

Lian, Y., & Shyy, W. (2007). Laminar-turbulent transition of a low Reynolds number rigid or flexible airfoil. *AIAA journal*, 45(7), 1501-1513.

Lin, J. C., Robinson, S. K., McGhee, R. J., & Valarezo, W. O. (1994). Separation control on high-lift airfoils via micro-vortex generators. *Journal of aircraft*, 31(6), 1317-1323.

Lindstrom, A., Monastero, M., & Amitay, M. (2018). The flow physics of synthetic jets interaction with flow over a flapped airfoil. In *2018 Flow Control Conference* (p. 4019).

Maldonado, V., Peralta, N., Gorumlu, S., Ayele, W., & Santos, D. (2021). Aeroelastic response of a propulsive rotor blade with synthetic jets. *Journal of Fluids and Structures*, 107, 103411.

Manolesos, M., & Voutsinas, S. G. (2016). Experimental study of drag-reduction devices on a flatback airfoil. *Aiaa Journal*, 54(11), 3382-3396.

Mdouki, R. (2017, August). Passive Control of Laminar Bubble Separation for S809 Wind Turbine Airfoil via Slot. In *Proc. 23rd French Congress of Mechanics, Lille, France*.

Metzinger, C. N., Chow, R., Baker, J. P., Cooperman, A. M., & Van Dam, C. P. (2018). Experimental and computational investigation of blunt trailing-edge airfoils with splitter plates. *AIAA Journal*, 56(8), 3229-3239.

Mishra, A., & Baeder, J. D. (2016). Coupled aeroelastic prediction of the effects of leading-edge slat on rotor performance. *Journal of Aircraft*, 53(1), 141-157.

Moghaddam, T., & Neishabouri, N. B. (2017, October). On the active and passive flow separation control techniques over airfoils. In IOP Conference Series: Materials Science and Engineering (Vol. 248, No. 1, p. 012009). IOP Publishing.

Mohamed, M. A. R., Guven, U., & Yadav, R. (2019). Flow separation control of NACA-2412 airfoil with bio-inspired nose. *Aircraft Engineering and Aerospace Technology*, 91(7), 1058-1066.

Mohamed, O. S., Ibrahim, A. A., Etman, A. K., Abdelfatah, A. A., & Elbaz, A. M. (2020). Numerical investigation of Darrieus wind turbine with slotted airfoil blades. *Energy Conversion and Management: X*, 5, 100026.

Mohamed, W. M. W., Ravindran, N. P., & Rajendran, P. (2021). A CFD Simulation on the Performance of Slotted Propeller Design for Various Airfoil Configurations. *CFD Letters*, 13(3), 43-57.

Nakagawa, T. (1992). Effects of a small airfoil-shaped splitter plate on vortex shedding from a square prism at subsonic Mach numbers. *Acta mechanica*, 91(1), 11-25.

Narayanan, S., Chaitanya, P., Haeri, S., Joseph, P., Kim, J. W., & Polacsek, C. (2015). Airfoil noise reductions through leading edge serrations. *Physics of Fluids*, 27(2), 025109.

Ni, Z., Dhanak, M., & Su, T. C. (2019). Improved performance of a slotted blade using a novel slot design. *Journal of Wind Engineering and Industrial Aerodynamics*, 189, 34-44.

Ohashi, M., Morita, Y., Hirokawa, S., Fukagata, K., & Tokugawa, N. (2020). Parametric study toward optimization of blowing and suction locations for improving lift-to-drag ratio on a Clark-Y airfoil. *Journal of Fluid Science and Technology*, 15(2), JFST0008-JFST0008.

Ol, M., McCauliffe, B., Hanff, E., Scholz, U., & Kähler, C. (2005, June). Comparison of laminar separation bubble measurements on a low Reynolds number airfoil in three facilities. In *35th AIAA fluid dynamics conference and exhibit* (p. 5149).

Olsman, W. F. J., & Colonius, T. (2011). Numerical simulation of flow over an airfoil with a cavity. *AIAA journal*, 49(1), 143-149.

Ozkan, G. M. (2021). Control of the flow in the near wake of an airfoil at full stall using fixed trailing-edge plates. *Experimental Thermal and Fluid Science*, 122, 110296.

Pan, C., Yu, D., & Wang, J. (2011). Dynamical mode decomposition of Gurney flap wake flow. *Theoretical and Applied Mechanics Letters*, 1(1), 012002.

Pröbsting, S., & Yarusevych, S. (2021). Airfoil flow receptivity to simulated tonal noise emissions. *Physics of Fluids*, 33(4), 044106.

Seshagiri, A., Cooper, E., & Traub, L. W. (2009). Effects of vortex generators on an airfoil at low Reynolds numbers. *Journal of Aircraft*, 46(1), 116-122.

Singh, M. K., Dhanalakshmi, K., & Chakrabartty, S. K. (2007). Navier-Stokes analysis of airfoils with Gurney flap. *Journal of Aircraft*, 44(5), 1487-1493.

Song, K. (2020, April). Numerical Investigation of Flatback Airfoils Drag and Noise Reduction by a Splitter Plate. In *IOP Conference Series: Earth and Environmental Science* (Vol. 495, No. 1, p. 012081). IOP Publishing.

Storms, B. L., & Jang, C. S. (1994). Lift enhancement of an airfoil using a Gurney flap and vortex generators. *Journal of Aircraft*, 31(3), 542-547.

Strike, J., Hind, M., Saini, M., Naughton, J., Wilson, M., & Whitmore, S. (2010). Unsteady surface pressure reconstruction on an oscillating airfoil using the wiener deconvolution method. In *27th AIAA aerodynamic measurement technology and ground testing conference* (p. 4799).

Suzuki, S., & Ishii, K. (2000). Numerical simulation of the acoustic control of separation over 2D airfoils. In *6th Aeroacoustics Conference and Exhibit* (p. 1996).

Taherian, G., Nili-Ahmadabadi, M., Karimi, M. H., & Tavakoli, M. R. (2017). Flow visualization over a thick blunt trailing-edge airfoil with base cavity at low Reynolds numbers using PIV technique. *Journal of Visualization*, 20(4), 695-710.

Timpe, A., Zhang, Z., Hubner, J., & Ukeiley, L. (2013). Passive flow control by membrane wings for aerodynamic benefit. *Experiments in fluids*, 54(3), 1-23.

Tousi, N. M., Coma, M., Bergadà, J. M., Pons-Prats, J., Mellibovsky, F., & Bugeda, G. (2021). Active flow control optimisation on SD7003 airfoil at pre and post-stall angles of attack using synthetic jets. *Applied Mathematical Modelling*, 98, 435-464.

U.S. Centennial of flight commission, 2021. <https://www.centennialofflight.net/>.

Venkatesh, S., Vimal, S. R., Manigandan, S., & Gunasekar, P. (2020). Aerodynamic Investigation on Performance Enhancement of NACA 2412 Airfoil with Suction Assistance. *International Journal of Vehicle Structures & Systems*, 12(1), 74-78.

Viieru, D., Tang, J., Lian, Y., Liu, H., & Shyy, W. (2006, January). Flapping and flexible wing aerodynamics of low Reynolds number flight vehicles. In *44th AIAA Aerospace Sciences Meeting and Exhibit* (p. 503).

Wang, J. J., Lan, S. L., & Chen, G. (2000). Experimental study on the turbulent boundary layer flow over riblets surface. *Fluid dynamics research*, 27(4), 217-229.

Wang, J., & Feng, L. (2019). Flow control techniques and applications. Cambridge University Press.

Winzen, A., Roidl, B., Klän, S., Klaas, M., & Schröder, W. (2014). Particle-image velocimetry and force measurements of leading-edge serrations on owl-based wing models. *Journal of Bionic Engineering*, 11(3), 423-438.

Xu, F., Zhong, S., & Zhang, S. (2018). Vortical structures and development of laminar flow over convergent-divergent riblets. *Physics of Fluids*, 30(5), 051901.

Yarusevych, S., Sullivan, P. E., & Kawall, J. G. (2007). Effect of acoustic excitation amplitude on airfoil boundary layer and wake development. *AIAA journal*, 45(4), 760-771.

Yee, K., Joo, W., & Lee, D. H. (2007). Aerodynamic performance analysis of a Gurney flap for rotorcraft application. *Journal of Aircraft*, 44(3), 1003-1014.

Zhu, H., Hao, W., Li, C., & Ding, Q. (2019). Numerical study of effect of solidity on vertical axis wind turbine with Gurney flap. *Journal of Wind Engineering and Industrial Aerodynamics*, 186, 17-31.

Zhu, J., Zhu, M., Zhang, T., Zhao, H., & Wang, C. (2021). Improvement of the power extraction performance of a semi-active flapping airfoil by employing two-sided symmetric slot airfoil. *Energy*, 227, 120458.

Zong, H., & Kotsonis, M. (2018). Formation, evolution and scaling of plasma synthetic jets. *Journal of Fluid Mechanics*, 837, 147-181.