CFD Simulation and Wind Tunnel Test of NACA 4412 Airfoil Wing Section at Different Angles of Attack

by

Ahmad Khairuddin Bin Ahmad Kamal

Dissertation submitted in partial fulfillment of
The requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JANUARY 2008

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan.

CERTIFICATION OF APPROVAL

CFD Simulation and Wind Tunnel Test of NACA 4412 Airfoil Wing Section at Different Angles of Attack

by

Ahmad Khairuddin bin Ahmad Kamal

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

____

(AP Dr. Hussain H. Al-Kayiem)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK

January 2008

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

AHMAD KHAIRUDDIN BIN AHMAD KAMAL

ABSTRACT

Airfoil can be defined as a body shape so that lift can be generated as fluid flow through it. The study of thin aerofoil by changes in a few characteristic such as different angle of attack, α would provide changes in lift, drag, and pitching moment coefficients. In order to identify the changes, Computational Fluid Dynamic (CFD) simulation has been applied by using ANSYS, FLUENT and GAMBIT software and followed by experiments conducted in a low speed wind tunnel to obtain measurement results. Shape of the body is important in achieving the desired result. NACA 4412 airfoil, which is high lift wing type, was chosen as the airfoil shapes in the present work. It was identified along the entire project. The aerodynamics characteristics of this airfoil have been shown by the plot of the coefficient graph lift and drag. This results obtained by both CFD and wind tunnel test. Although there is a little differences between results of these two methods, but, the general trend is the same. The results were compared with previous experimental results and they are in good agreement. It can be concluded that Reynolds Number values are affecting the characteristic of the airfoil and this is the reason for the small differences of recent results from others. However, a few recommendations have been outlined for further investigation.

ACKNOWLEDGEMENTS

The greatest gratefulness goes to Allah for His blessings, kindness and never ending love for nourishing me with health and hope throughout my journey of life. My sincere appreciation goes to Universiti Teknologi PETRONAS (UTP) for giving me a great opportunity to pursue further education in Bachelor of Engineering, Mechanical Engineering. I am very grateful for this golden chance that has contributed a lot in developing my interpersonal skills in both technical and non-technical field.

Many individuals have contributed to the continued development and refinement of this project. To these wonderful people, I express my endless appreciation and gratitude. Special thanks to AP Dr. Hussain H. Al-Kayiem my Final Year Project supervisor for his tremendous support, advices and guidance throughout this course. Not forget my dear colleague Mr Muhd Nuh Aizat b Muhd Daut who was fabricating the airfoil for experimental wind tunnel test.

Last but not least, special recognition is due to my beloved mother – Seleni bt Merah and grandparent – Merah bin Ismail and Fatimah bt Bakar for their endless love and support through thick and thin in leading life.

Thank you.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Background

The idea of airfoil basically developed in order to minimize the average pressure at the upper surface while maximizing it at the lower surface. The concept is designed based on Bernoulli Principle which the pressures are inversely proportional with the velocity [1]. When it comes to the airfoil, lift will be generated when the flow of velocity at top surface is higher and lead to decrease the pressure.

In order to study the characteristics of airfoil, NACA 4412 model was selected as the shape of body. This 2-D section was introduced by Abot and Von Doenhoff (1959) and also by Ladson and Brooks Jr. (1975) with the purpose of airfoil geometries could be easily studied [2]. In 1958, NACA was change into National Aeronautics and Space Administration (NASA).

Angle of attack, α, became the main character to be tested during both simulation and experiments. The definition of angle of attack was described as "the angle at which the wing is inclined relative to the air flow" (R.H. Barnard and D.R. Philpott, 1995, p.9) [3]. The adjustment to the angle of attack would lead spectacular change in lift, C_L drag, C_D, pitching moment, C_M. For NACA 4412, it can be categorized as high lift wing.

Nowadays, CFD become the most powerful tool to simulate the aerodynamic characteristics of the airfoil wing section. Many parties try to study the characters of the wing and come up with a few optimizations to that wing according to its purpose. At the end of this project, the performance of this wing should be increase by a few adjustments and proved by the experiment that will be conducted in low speed wind tunnel. Refer to Wikipedia [4]; wind tunnel can be defined as a research tool developed to assist with studying the effects of air moving over or around solid objects. Ways that wind-speed and flow are measured in wind tunnels.

1.2 Problem Statement

The aerodynamic characteristics of thin airfoils are required to be studied at different operation conditions. Any modification to the airfoil would lead to dramatic change in the lift, drag and pitching moment coefficients. Using the CFD simulation technique with wind tunnel test for verification represents powerful tool to investigate the aerodynamic characteristics of the wings.

Figure 1.1: The example of used NACA 4412 wing section in aircraft industry

Most of the jet fighter nowadays use this kind of airfoil due to high lift characteristic perform by this type of wing section. So that, the analysis for this airfoil should be useful to identify why it has this character and the relationship between experimental and the true wing section used for aircraft industry.

1.3 Objectives and Scope of Study

The CFD simulation will discover the aerodynamic characteristics of the NACA 4412 airfoil and proved by the experiment using low speed wind tunnel. Hence, the project targets are as follows:

- To apply the CFD software to simulate and evaluate the aerodynamic characteristics of NACA 4412 airfoil at different angle of attack.
- To perform experimental investigation on the aerodynamic characteristics of the wing model in the subsonic speed wind tunnel at different angle of attack

CHAPTER 2

LITERATURE REVIEW AND THEORY

Referring to [5], the four digit numbering system is very important. By that way, the airfoil can be distinguished based on its criteria.

- First integer: Maximum value of the mean-line ordinate in percent chord.
- Second integer: Distance from leading edge to the location of maximum chamber in tenths of the chord.
- Last two integers: Section thickness in percent of the chord.

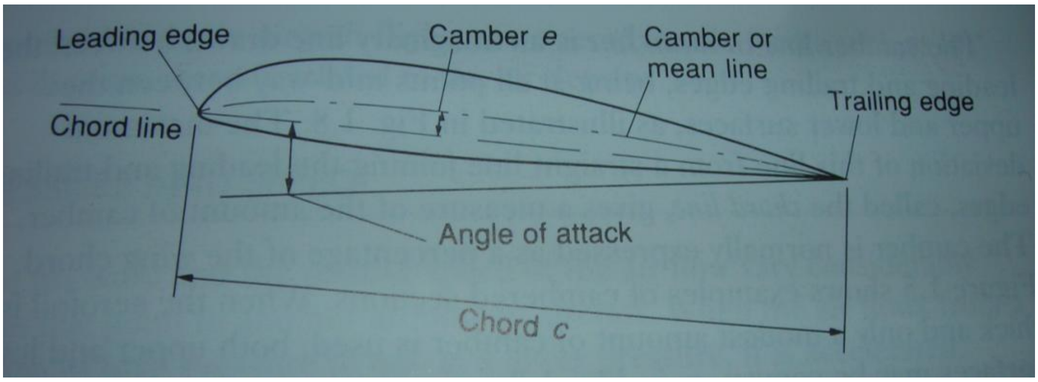

Figure 2.1: Cambered aerofoil

The degree of chamber is usually expressed as a percentage of the chord, (e/c) x 100% [3]. For wing section 4412, it has 4% chamber at 0.4 of the chord from leading edge and is 12 % thick. This NACA foil definition is very ideas to engineers because its apply ht concept of chamber and thickness to define the shape [2].

The principles that will be evaluated which are lift, C_L drag, C_D, pitching moment, C_M coefficients also has their own theory.

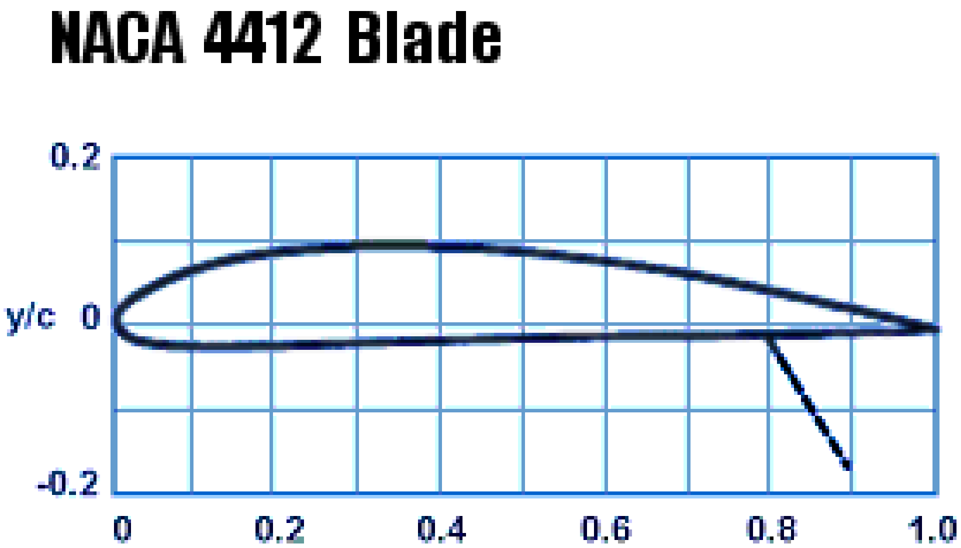

Figure 2.2: The sketch of the shape NACA 4412 section

2.1 Lift, Drag and Pitching Moment

Dynamic head, $q = 1/2\rho V^2$ N/m^2

Dynamic pressure [6] has the units of pressure. In addition, let S be a reference area and l be a reference length. The dimensionless force and moment coefficients are defined as follows:

Coefficient of lift, $C_L = \dfrac{L}{(q)(S)}$

Coefficient of drag, $C_D = \dfrac{D}{(q)(S)}$

Moment coefficient, $C_M = \dfrac{M}{(q)(S)(l)}$

2.2 Turbulence

Mechanically, turbulence is common near the ground as wind blowing over or around buildings create eddies. The turbulence parameter will be changing according to the velocity of the wind. By the way, most turbulence involves eddies. They are examples of the "random fluctuations" in "instantaneous velocities" in the scientific definition.

In fluid dynamics [7], turbulence or turbulent flow is a flow regime characterized by disordered property changes. This includes low momentum diffusion, high momentum convection, and rapid variation of pressure and velocity in space and time. Flow that is not turbulent is called laminar flow. The dimensionless Reynolds number characterizes whether flow conditions lead to laminar or turbulent flow. For example is for pipe flow, a Reynolds number above about 2300 will be turbulent.

CHAPTER 3

METHODOLOGY

3.1 Procedures Identification

Initially, the project started with research and brief analysis based on books, journals, technical papers, previous student's reports, thesis and articles obtained from both the Information Resource Centre (IRC) and internet. Also, some interviews and consultation sessions were held with the lecturers and related personnel on the project overview. To understand the concept as a whole, a site visit to the wind tunnel contain airfoil model might be helpful. The full understanding of the airfoil and its principle especially for NACA 4412 is necessary to be implemented in certain simulations and experiments in order to get the desire result. Then the different angle of attack will be used as the parameter to evaluate the changes of aerodynamic characteristics of the airfoil and the final outcome will be chosen as the end result of the project. Once the desired results are achieved by simulate with ANSYS, FLUENT, and GAMBIT software, the specimen will be tested in the subsonic wind tunnel to observe the flow distribution and obtain the data required previously. The familiarization of the ANSYS, FLUENT, and GAMBIT software will be done by using online tutorials, reference books and self attempt while the experiment using wind tunnel will be helped by the responsible technician. Finally, the project will be finalized based on data gathered and further enhancement will be implemented.

3.2 Flowchart

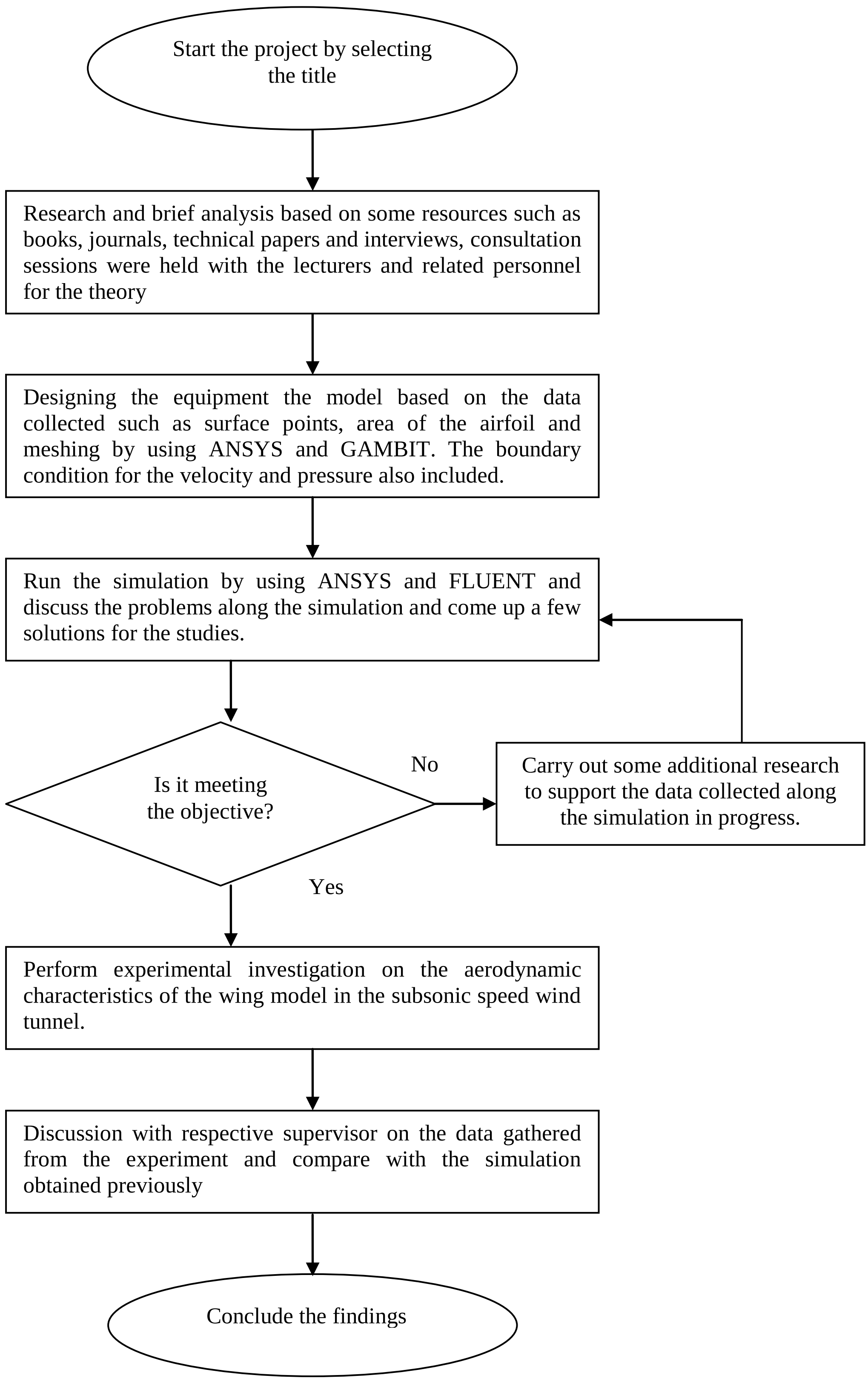

Figure 3.1: The example of flowchart used in the project

3.3 Tools/Equipment Required

The tools/equipment required in this project is ANSYS, FLUENT, and GAMBIT software. In addition, the subsonic speed wind tunnel also will be utilized to obtain the result required. In order to synchronize the data gathered, Microsoft Excel plays the major role to overcome the problems.

CHAPTER 4

RESULTS AND DISCUSSION

The finding of project are divided into the results obtained according to the software used in the project, start with ANSYS, GAMBIT and FLUENT and followed by the experimental outcome from the wind tunnel test.

4.1 Simulation Results by ANSYS

4.1.1 Surface points

The surface point which is 99 key points are generated through the result obtained from the surface points data that evaluate the airfoil characteristic for chord length 105mm. This length are used because the size of the model that will be test in the wind tunnel also same with it. This original surface points was introduced by Al-Kayiem, (1989) [8] for chord with length 250mm. Below is the surface points for chord length equal to 105mm in turn to provide the shape of airfoil with NACA 4412 characteristic. Refer Appendix 1.

4.1.2 Designing and Modeling

The first step to model the airfoil is by creating surface points refer from the data from the table. The coordinate are started from the origin and continue with the upper surface before proceeding with the lower surface. Refer figure 4.1.1 for the result of surface points created.

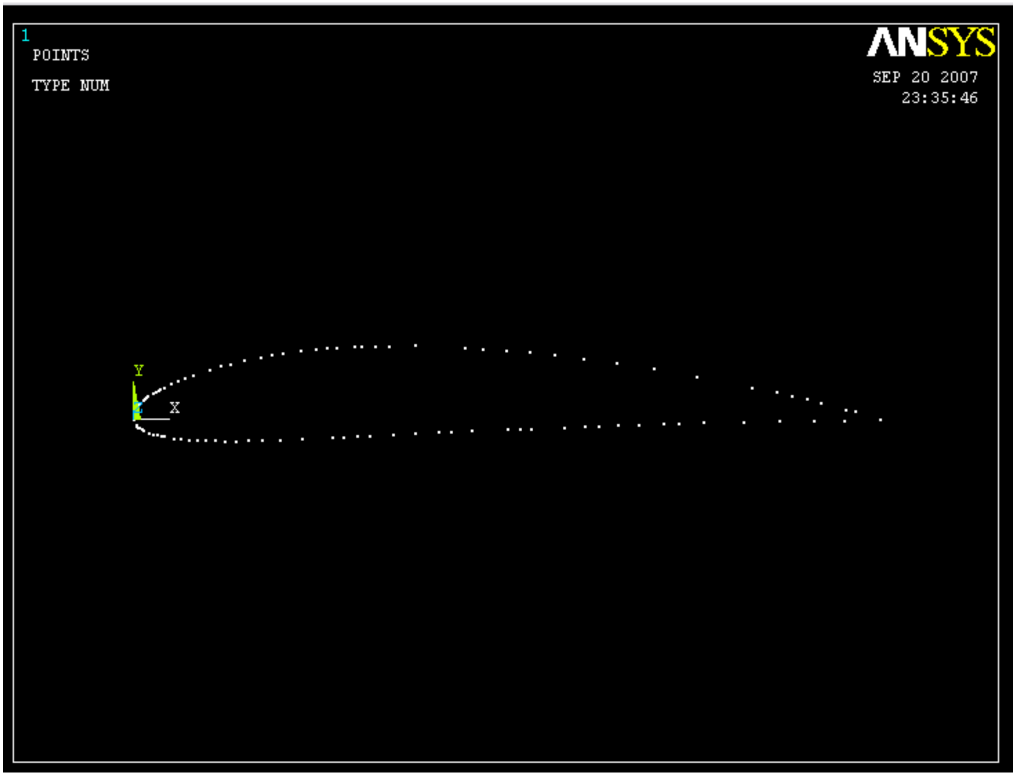

Figure 4.1.1: Surface points geometry by using ANSYS

For more details for the number of the points created, refer to the figure 4.1.2. It shows all the key points generated by the software before connected by the lines in turn to provide the area of the airfoil. The lines connected all the key points and provide an area called airfoil with NACA 4412 characteristic. Refer figure 4.1.3 for details of the lines. By connecting all the surface points, the model and shape of NACA 4412 airfoil section can be seen and prepared for further investigation in ANSYS software.

Figure 4.1.2: Key points on the surface points

Figure 4.1.3: Lines connect all the surface points

The meshing procedure use free mesh option with element edge is 20. The area along the airfoil is 300mm x 300mm. But, that area is not the wind tunnel area. It was chosen by using trial and error procedure. Refer figure 4.1.4 for the result of the meshing.

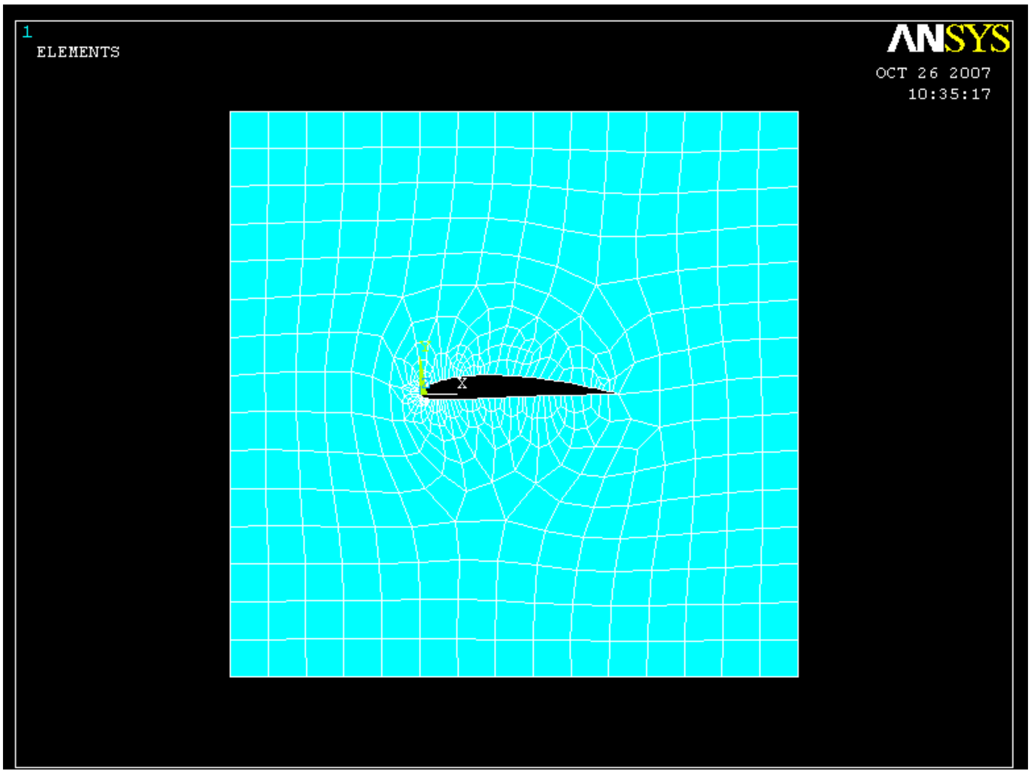

Figure 4.1.4: Mesh through all surface points by using ANSYS

4.1.3 Possible method to get lift and drag

Take around 20 nodes each above and below in order to get the static pressure onto the body of airfoil. By putting all the value in Microsoft Excel like table in Appendix 2.

4.1.4 Possibility to apply the angle of attack

There are 2 ways proposed for the method. For the first option, the tunnel design has being change to provide elevation onto the airfoil. If refer to the above diagram, the elevation around 4º has been set to the airfoil. To obtain the data, the simulation will be run to get the pressure distribution and lead to the results of lift, drag and pressure distribution.

For the second option, the angle of attack obtained by differentiates the value of velocity on x-axis and y-axis to the line of boundary condition.

4.1.5 Findings

The boundary condition is very important characteristic in order to evaluate this simulation. For this first simulation, angle of attack of the velocity is 0º and the velocity 16630 mm/s. Refer figure 4.1.5 to check the place for the velocity and pressure boundary condition.

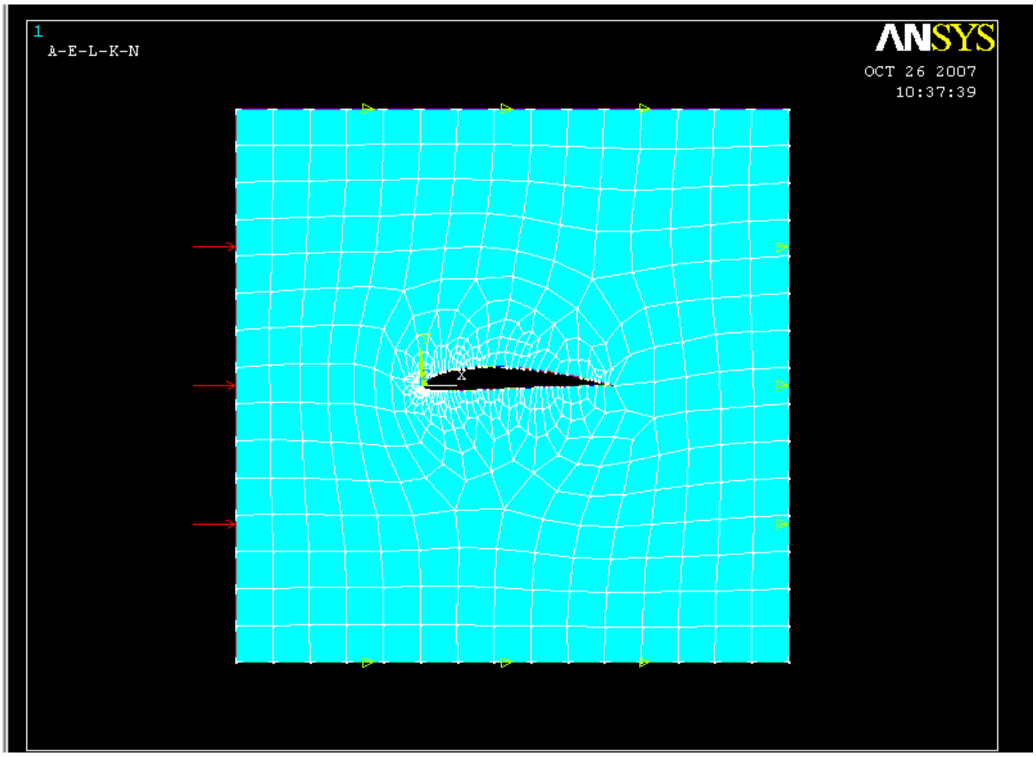

Figure 4.1.5: Put load at boundary condition

The value of velocity obtained from this equation;

$$Re_c = \frac{\rho_{air}\ V\alpha\ .\ C}{\mu_{air}} = 0.12 \text{ x } 106$$

Where,

Chord length, C = 105mm

Air density, $\rho_{air} = 1.23 \text{ x } 10^{-9}$ kg/mm^3

Air viscosity, $\mu_{air} = 1.79 \text{ x } 10^{-8}$ kg/s.mm

Therefore, velocity, Vα = 16630mm/s

From the fluid velocity distribution shown in figure 4.1.6, the area for meshing should be increase in order to get more specific distribution because the maximum velocity may be display around the top of the airfoil. However, it is just assumption and cannot be proof because the simulation with smaller area and increase the meshing tend to give the result divergent solution. Therefore, for the time being, the result below is the best solution of the simulation of NACA 4412 airfoil.

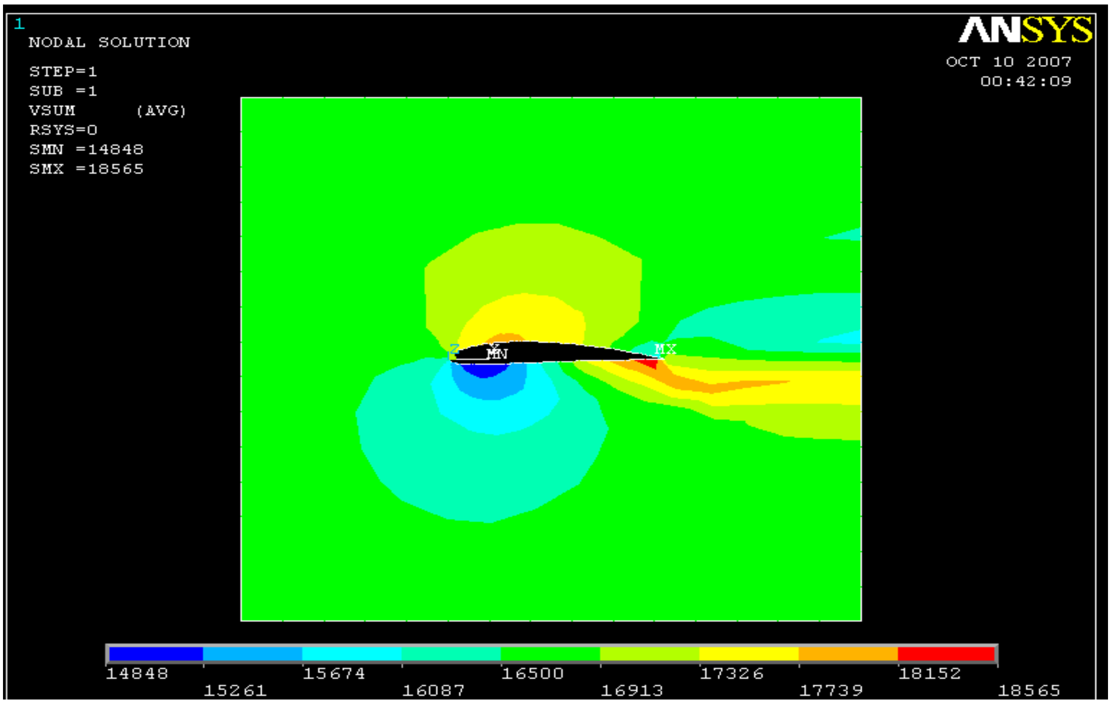

Figure 4.1.6: Fluid velocity along the airfoil

Coefficient of pressure should be useful to include into this analysis. The theory behind it will be explained by equation below.

$$\text{Coefficient of pressure, } C_P = \frac{\text{Pst model} - \text{Ps ref}}{\frac{1}{2}\,\rho\, V_\alpha}$$

Where,

Pst model = The static pressure of the surface

Ps ref = The static pressure of the free stream

Density, ρ = Density of air

Velocity, V_α = Velocity of air

The static wall pressure at surface data are obtained form the list of result of pressure distribution shown in figure 11 below and figure 12 for details of the nodes.

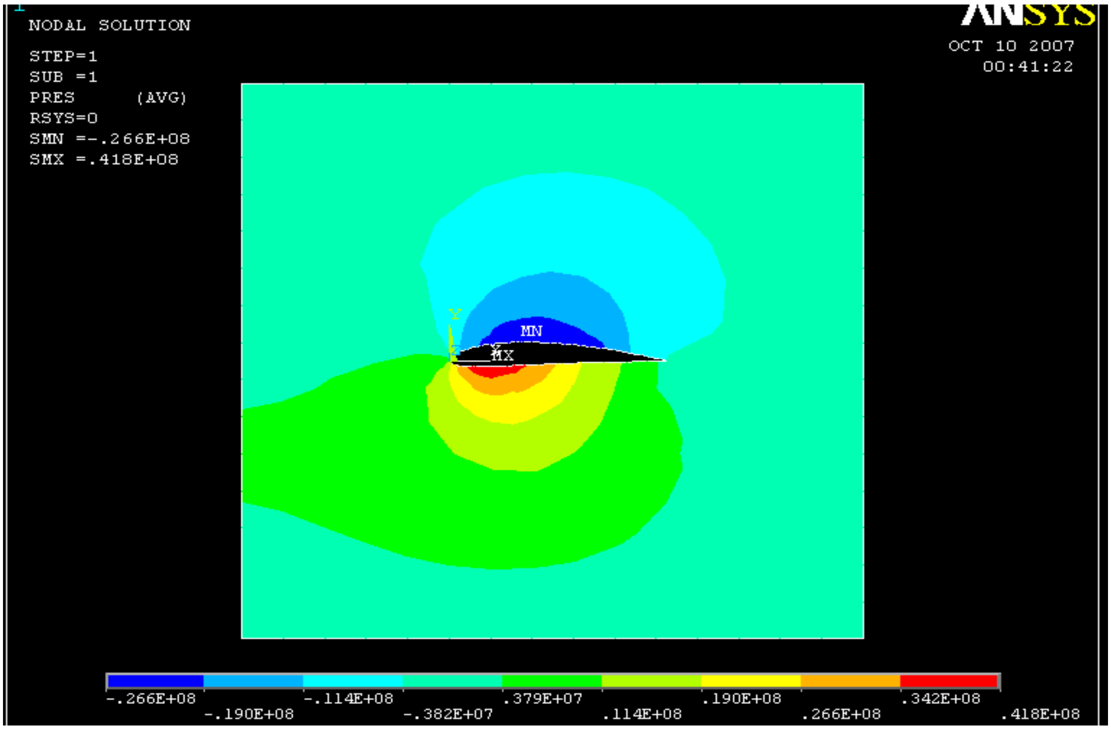

Figure 4.1.7: Pressure distribution along the airfoil

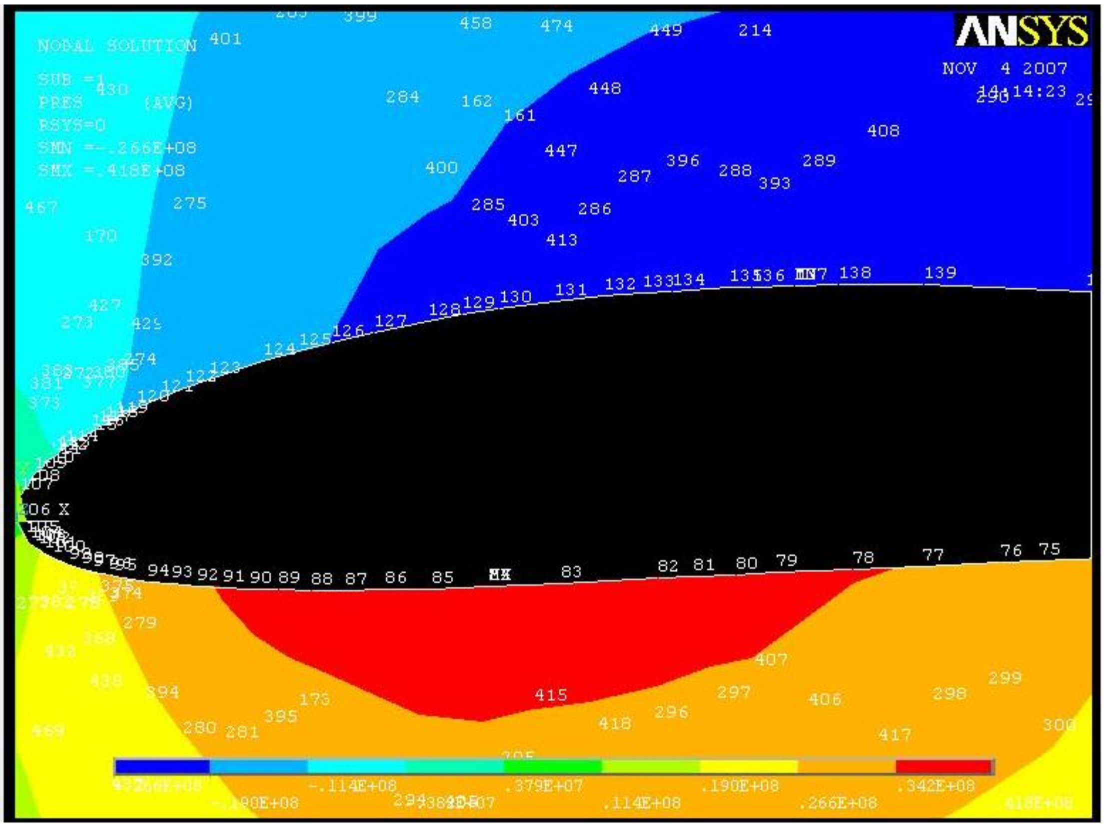

Figure 4.1.8: The nodes of selected key points from pressure distribution figure

There are differences between the pressure at the upper surface and the lower surface. The upper surfaces indicate negative sign while the lower surfaces indicate positive sign. The differences between the pressure from different condition occur due to pressure at the lower surface create back pressure on the upper surface.

By putting the value of Ps ref which is pressure at atmosphere that equal to 1.01325 x 10^{-4} N/mm^3, density equal to 1.23 x 10^{-9} kg/mm^3 and velocity obtained equal to 16630 mm/s, the result gathered are as below.

Table 4.1.1: The coefficient of pressure at selected points

Upper surface

	Node	Static Pressure at Wall	Coefficient of Pressure
1	110	-6.94E+05	-4079373.402
2	119	-1.13E+07	-66461866.64
3	129	-2.36E+07	-138538655.7
4	133	-2.50E+07	-147052171.5
5	137	-2.66E+07	-156635756.3
6	141	-2.48E+07	-145735163.5
7	145	-2.23E+07	-131224557.8
8	149	-1.24E+07	-72747052.01
9	153	-4.46E+06	-26227861.01

Lower surface

	Node	Static Pressure at Wall	
1	102	1.83E+07	107336149.8
2	97	2.60E+07	152790563.4
3	91	3.49E+07	205171027.8
4	84	4.18E+07	245962997.9
5	80	3.73E+07	219076045
6	74	2.85E+07	167495192.6
7	69	1.92E+07	113133336.7
8	64	1.37E+07	80737292.34
9	158	6.31E+06	37074362.39

Figure 4.1.9 below display how to get the static pressure for a few data which required in order getting the results of the objective of this analysis. The pressure will be used to evaluate the Coefficient lift, C_L, Coefficient drag, C_D and Coefficient of moment, C_M by using the equation stated in the theory as a back up plan if that data cannot get automatically.

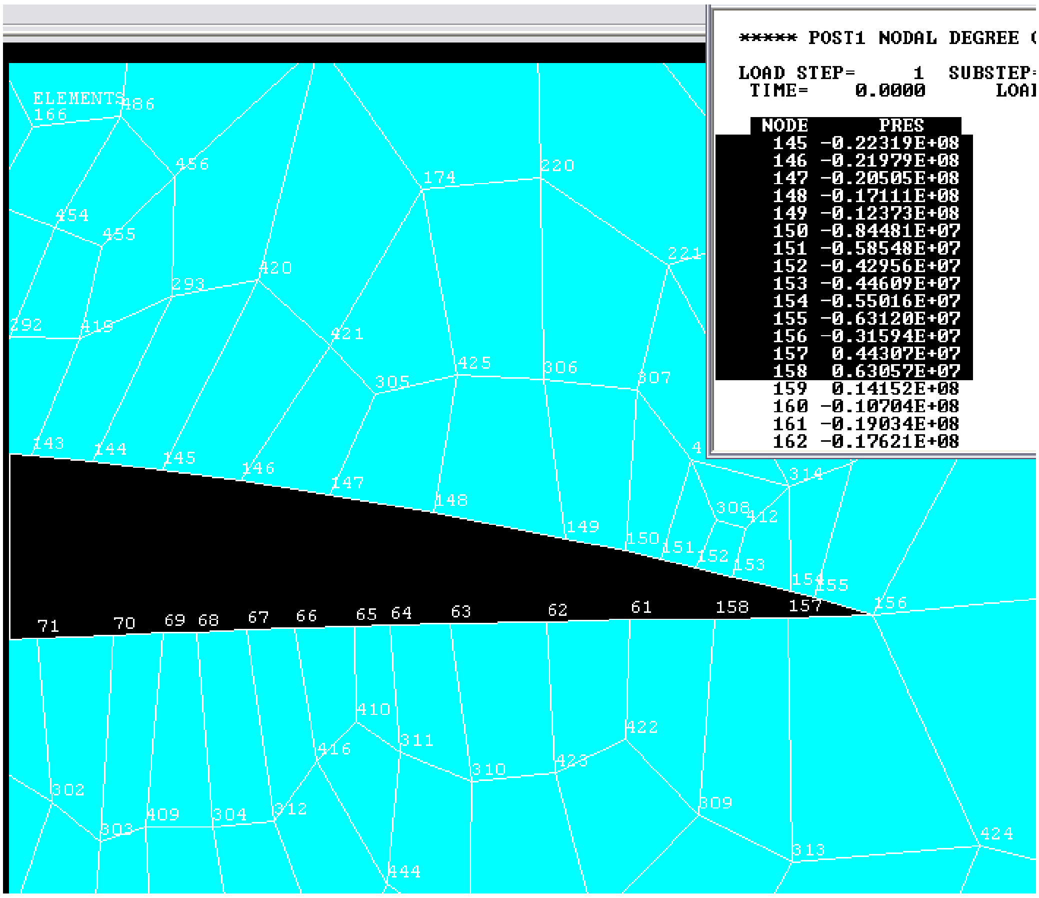

Figure 4.1.9: Pressure value at a few nodes

If this procedure is not supported the analysis, further investigation by using FLUENT software should be useful tool to fulfilling the objective of this analysis.

The first method being used again to evaluate the effect of 4 degree angle of attack, by using velocity 46 m/s, the pressure distribution of the airfoil can be evaluate as showed in the figure 4.1.10. However, after using FLUENT to simulate the airfoil, it is noted that FLUENT is more compatible software in order to evaluate on CFD problems.

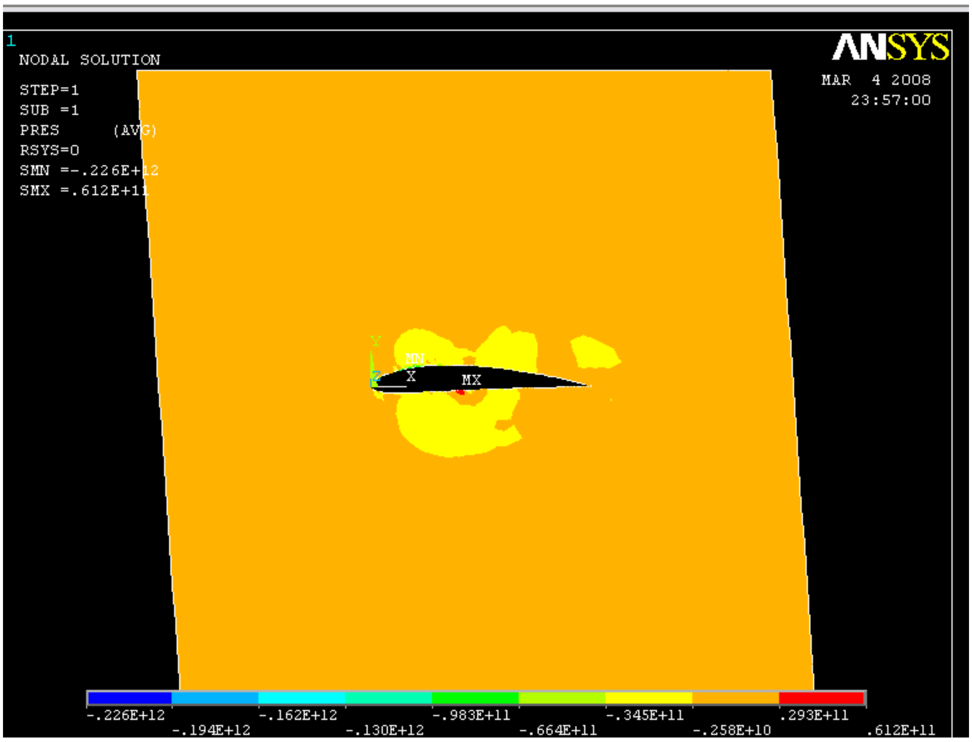

Figure 4.1.10: Pressure distribution along the airfoil

For further coordinates of the wind tunnel adjusted to specify the need of different angle of attack, refer to the Appendix 3.

4.2 Meshing by GAMBIT

GAMBIT software is one of the tool applicable in order to provide the geometry before get analyzed by FLUENT software. For this time, the surface nodes used are 61 upper and 61 lower [9], but still use NACA 4412 type of model. As we know, the more key points used in the geometry, the surface roughness become more refine. Refer to the Appendix 4 for the key points.

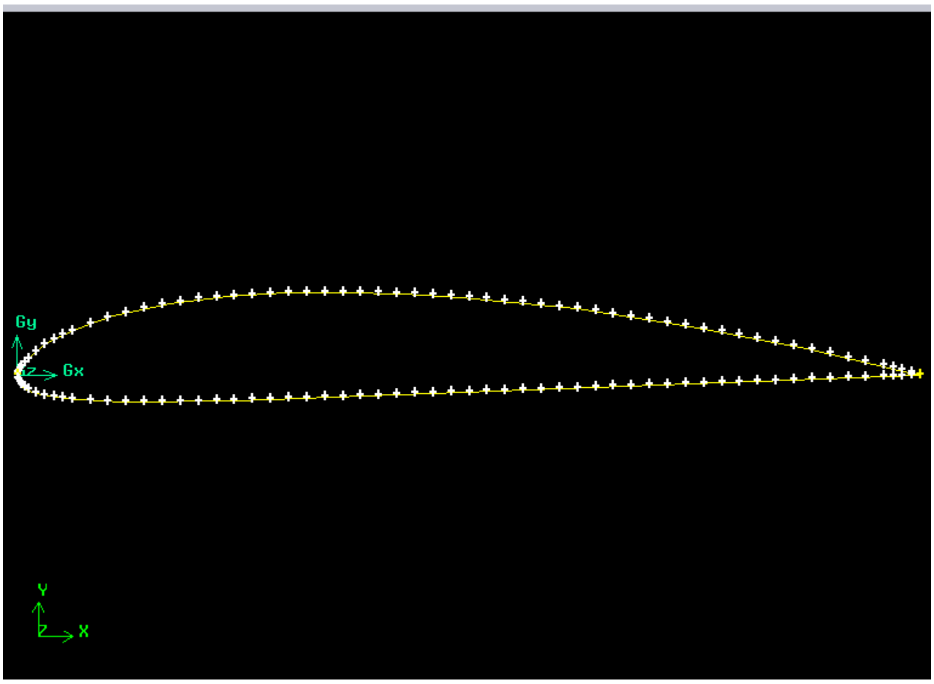

Figure 4.2.1: The vertices and edge of airfoil

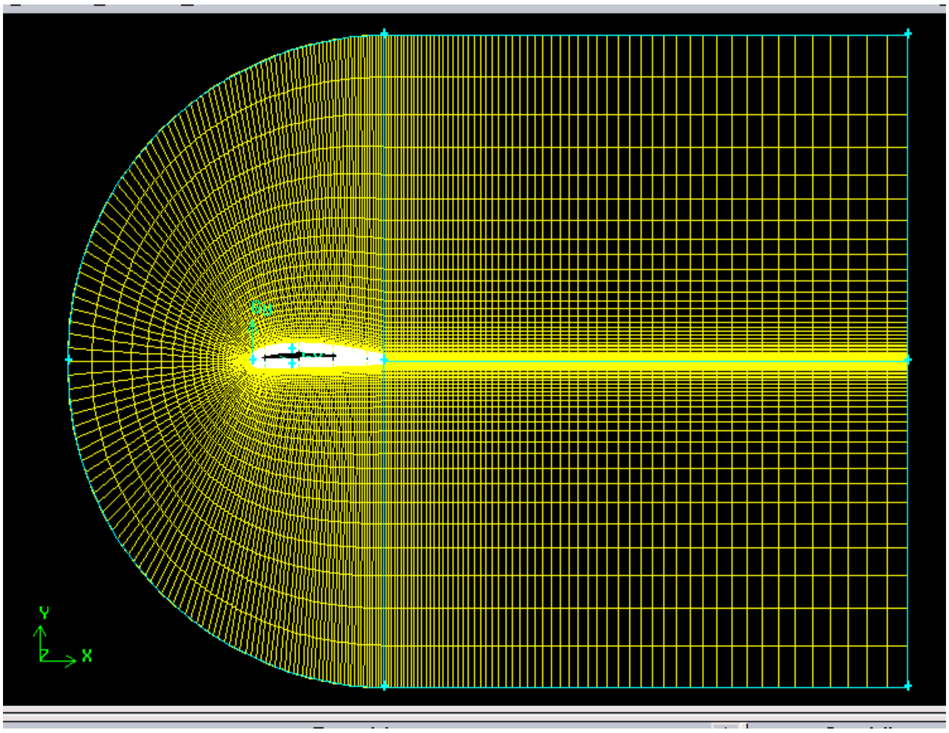

Figure 4.2.2: Done meshing

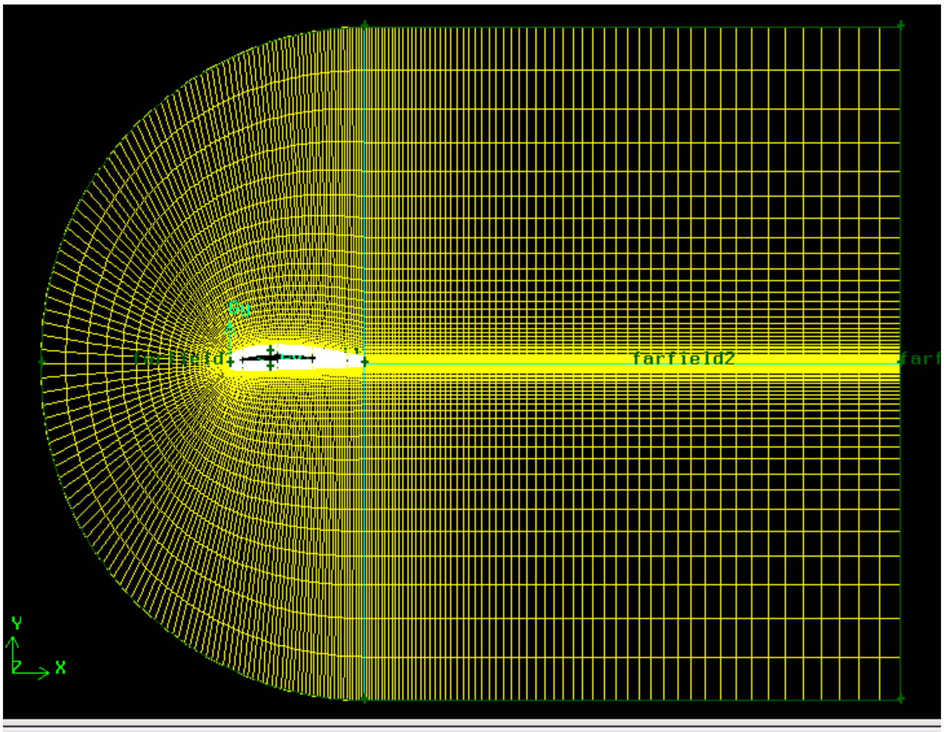

Figure 4.2.3: Ready to use by fluent after finishing grouping

4.3 Simulation Results by FLUENT

FLUENT is very useful tool to solve and analyze for CFD problems. In order to provide different angle of attacks, the boundary condition velocity was is set by using the second method of ANSYS previously by giving both velocity through x-direction and y-direction. Refer Table 4.3.1 below for the whole data of it.

Table 4.3.1: The x-direction velocity and y-direction velocity

velocity, v = 46m/s

x-component $\quad$ 46 x cos α

y-component $\quad$ 46 x sin α

angle of attack, α	cos α	sin α	x-velocity	y-velocity
-4	0.9976	-0.0698	45.888	-3.209
0	1	0	46	0
4	0.9976	0.0698	45.888	3.209
8	0.9903	0.1391	45.552	6.402
12	0.9781	0.2079	44.995	9.564
16	0.9613	0.2756	44.218	12.679
20	0.9397	0.342	43.226	15.733
22.5	0.9239	0.3827	42.498	17.603

4.3.1 Results for -4º angle of attack (example for FLUENT)

For the first try, the simulation was run by using the -4º angle of attack which means that the velocity inlet at the boundary condition was enter with 45.888m/s from the x-direction and -3.209m/s from the y-direction. When the FLUENT was running, the computer will define how many iteration that would provide the convergent solution that means the data obtained get stable and provided the coefficient needed. After that, once again FLUENT are used to identify the distribution of dynamic pressure, fluid velocity and the pressure coefficient at the airfoil.

The pressure coefficient can be defined as the dimensionless number that contain the values below than one (1). The numbers represent the relative pressures throughout a flow field around the airfoil by applying the fluid dynamics explanation. Every point at the airfoil contains its own pressure coefficient as discuss in the page 23 earlier. By referring to the figure 4.3.1 below, the upper and lower surface of the pressure coefficient showed by the plot of the graph according to its position start from the left. At the position 0m to 0.02m from left, the pressure coefficients higher at upper surface indicate by the plots which are up to around 1. While start from 0.02m to 0.105m, the pressure coefficient of the upper surface become lower than the lower surface.

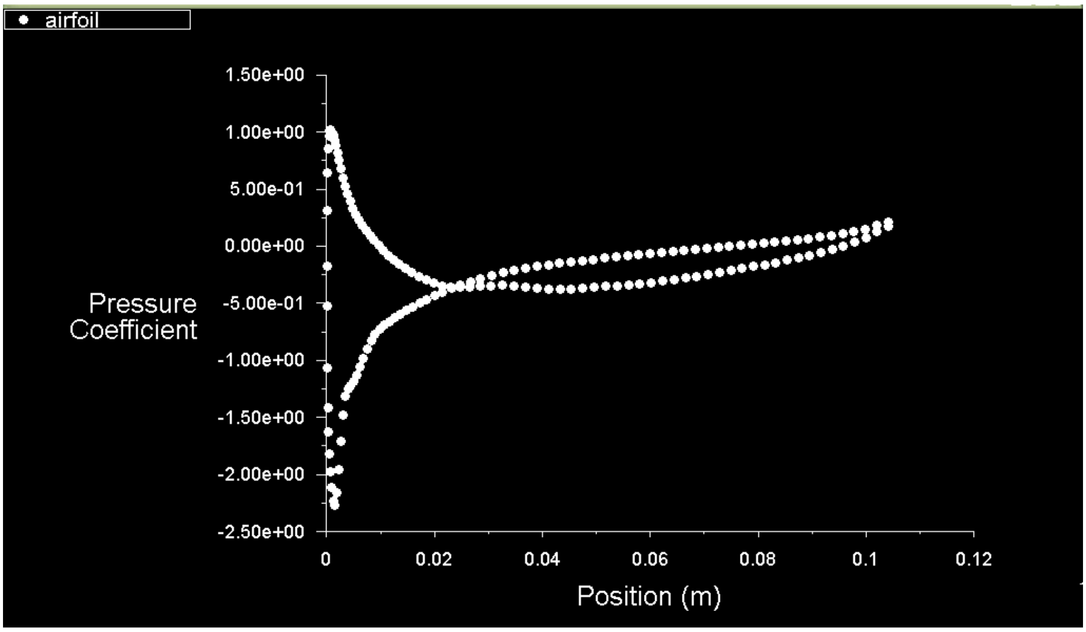

Figure 4.3.1: Pressure coefficient at the airfoil

Below is a few remarkable of the value of pressure coefficient, C_P

- $C_P = 0$; the pressure equal to free stream pressure.
- $C_P = 1$; the pressure is stagnation pressure the point is stagnation point.
- $C_P = -1$; significant in the design of gliders (fly with long period by not using an engine)

The static coefficient contour can be identified by referring the Figure 4.3.2. Most similar with the other CFD tools and indicate the highest pressure by red contour and the minimum pressure by blue contour. At the beginning of the upper surface, it showed by the maximum pressure contour. However, at the back of the lower surface mostly and upper surface a little shows also maximum pressure contour. Therefore, almost no lift created to the airfoil and it proved by the result of coefficient lift which is almost zero showed later.

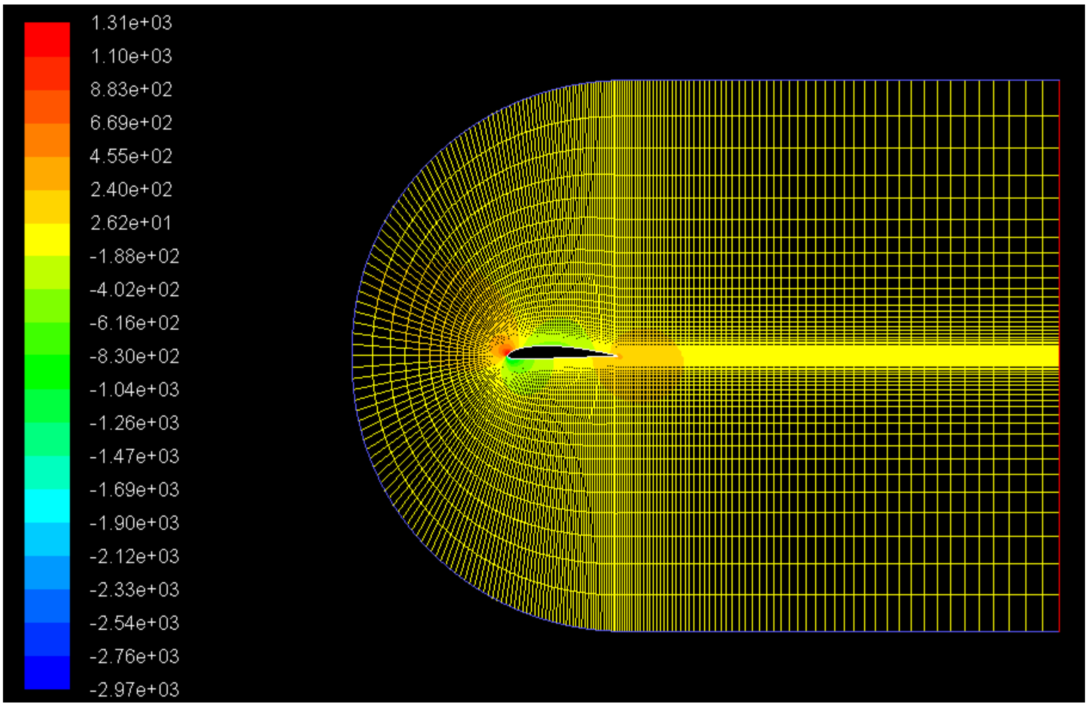

Figure 4.3.2: Static pressure distribution around the airfoil

The theory of dynamic pressure already explains in the Chapter 2.1 previously. Figure 4.3.3 below show the contour of the dynamic pressure, which means kinetic energy of a fluid particle. Dynamic pressure and kinetic energy both are proportional to the particle's mass and square of the velocity. Dynamic pressure also one of the terms in Bernoulli's equation, which is used in the evaluation of energy conservation for fluid in motion

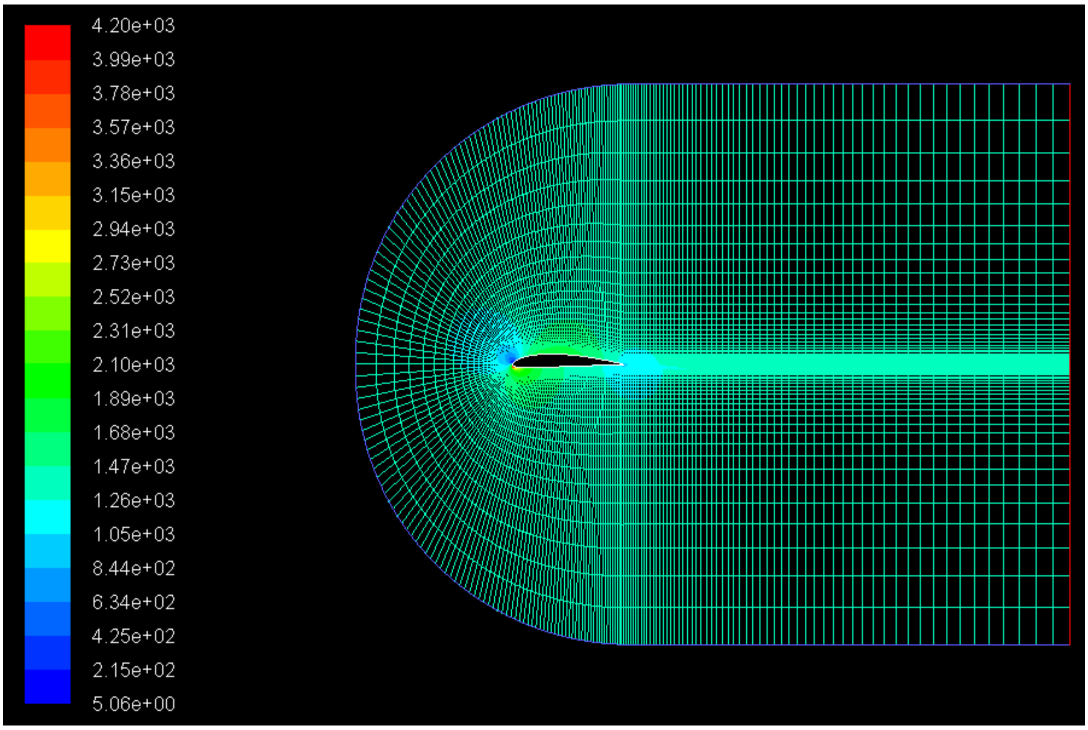

Figure 4.3.3: Dynamic pressure distribution around the airfoil

The characteristic of the velocity contour as usual will be different with the contour of static pressure contour. It proved by applying the Bernoulli Principle which stated that a region with high velocity will create low pressure. This principle will create lift to the certain shape, and also supported by NACA 4412 airfoil characteristic.

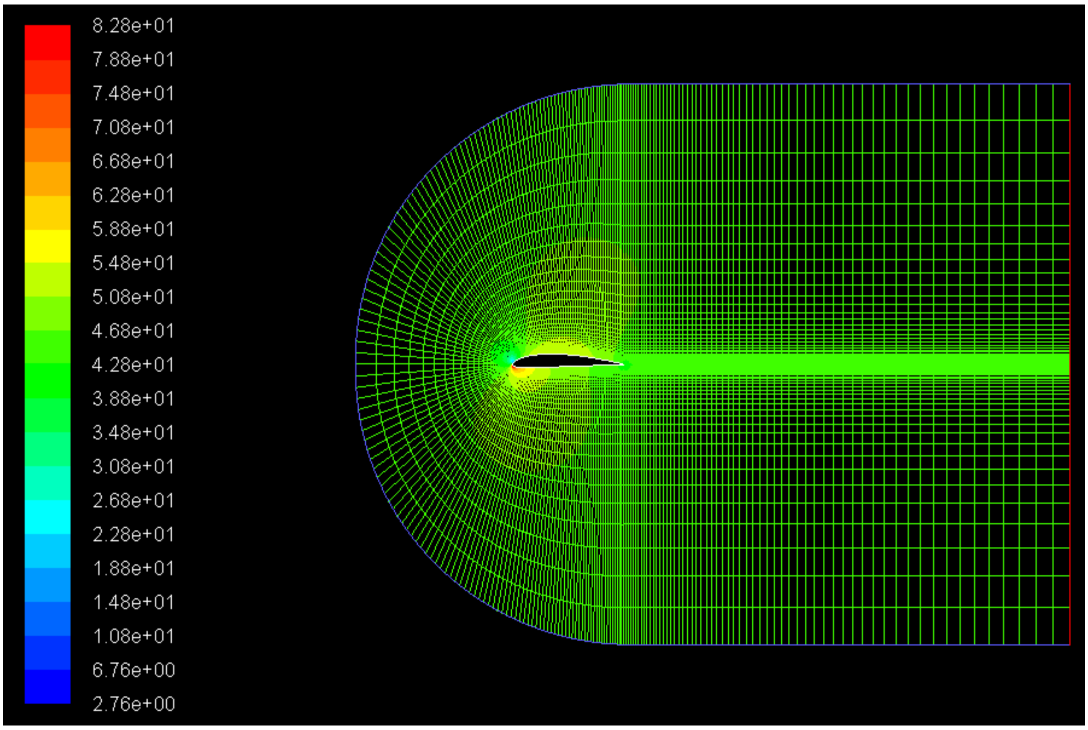

Figure 4.3.4: Contour of velocity magnitude around the airfoil

In order to get the stable and precise data, the FLUENT as the CFD tool has iterated 611 times to get the coefficient of all the forces. The iteration for every angle of attack is different and wider the angles of attack need more times to be iterated. Table 4.3.2 below shows the results for -4º angle of attack.

Table 4.3.2: Coefficient for -4º angle of attack of the simulation

Iteration	C_L	C_D	C_M
611	-6.4272e-03	9.5616e-05	7.5728e-04

*611 iterate before get stable data

Refer Appendix 5 for all the pressure coefficient, static pressure, and dynamic pressure contour and velocity magnitude of the other angle of attack.

4.3.2 Overall results for FLUENT

Table 4.3.3: Overall coefficient of the simulation

	CL	CD	CM
-4	-6.43E-03	9.56E-05	7.57E-04
0	4.19E-02	2.59E-04	2.10E-03
4	8.98E-02	1.16E-03	3.47E-03
8	1.40E-01	2.43E-03	4.84E-03
12	1.90E-01	4.33E-03	6.20E-03
16	2.39E-01	6.56E-03	7.47E-03
20	2.85E-01	8.85E-03	8.53E-03
22.5	3.11E-01	1.01E-02	8.97E-03

All the data gathered has been transfer into graph in order to evaluate the results for all coefficients of the forces. For the coefficient of lift, the coefficient keeps increase start from -4º until 20º angle of attack. However, it looks like to experienced stall (where the flow separates) after reaching the 22.5º angle of attack as shown in Figure 4.3.5. For the coefficient of drag, it also experiences the increasing of drag when the angles of attack become wider as shown in Figure 4.3.6. The different between lift and drag is, coefficient drag will be keep increase while the lift will turn down until certain points. Finally, the pitching moment coefficient, that causes the airfoil to pitch up or down also increase as shown in Figure 4.3.7. However, the plots of the graph quite similar with the plot of coefficient lift. It already mention that, lift on an airfoil acts at the point called center of pressure, which is affected with the pitching moment.

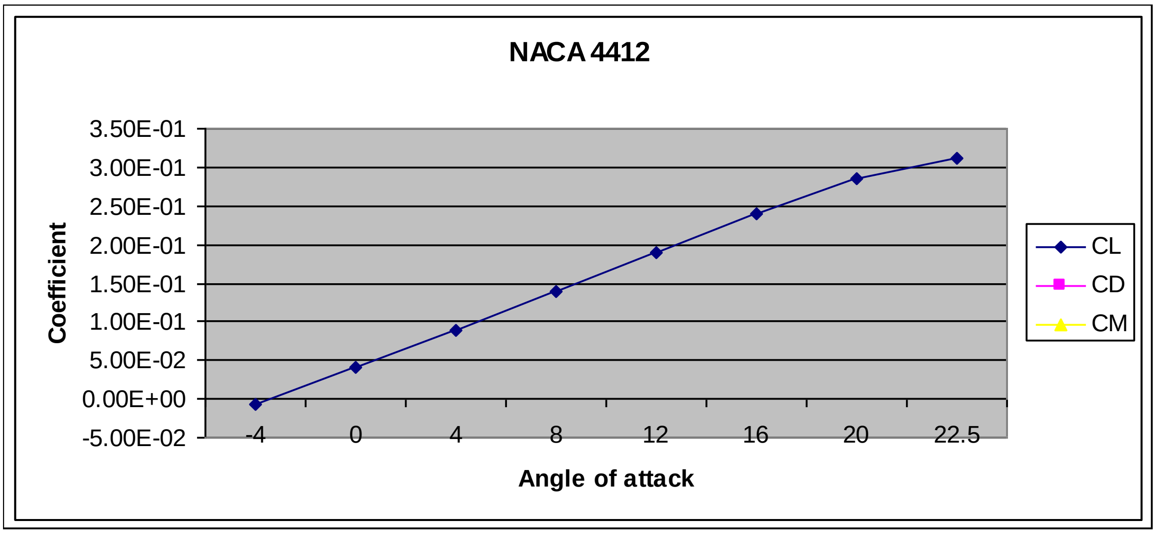

Figure 4.3.5: Lift coefficient vs. angle of attack

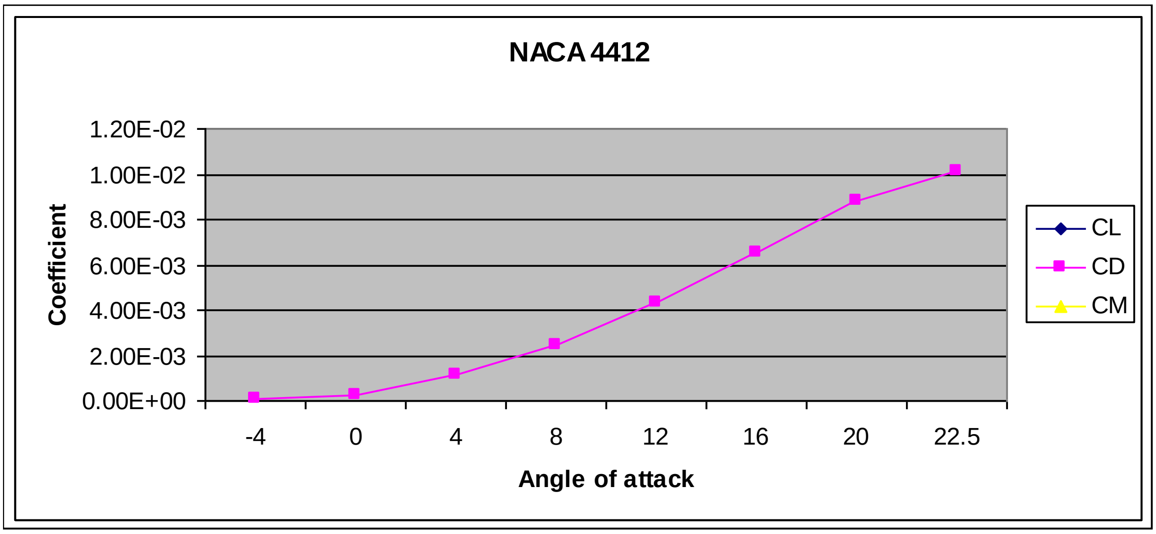

Figure 4.3.6: Drag coefficient vs. angle of attack

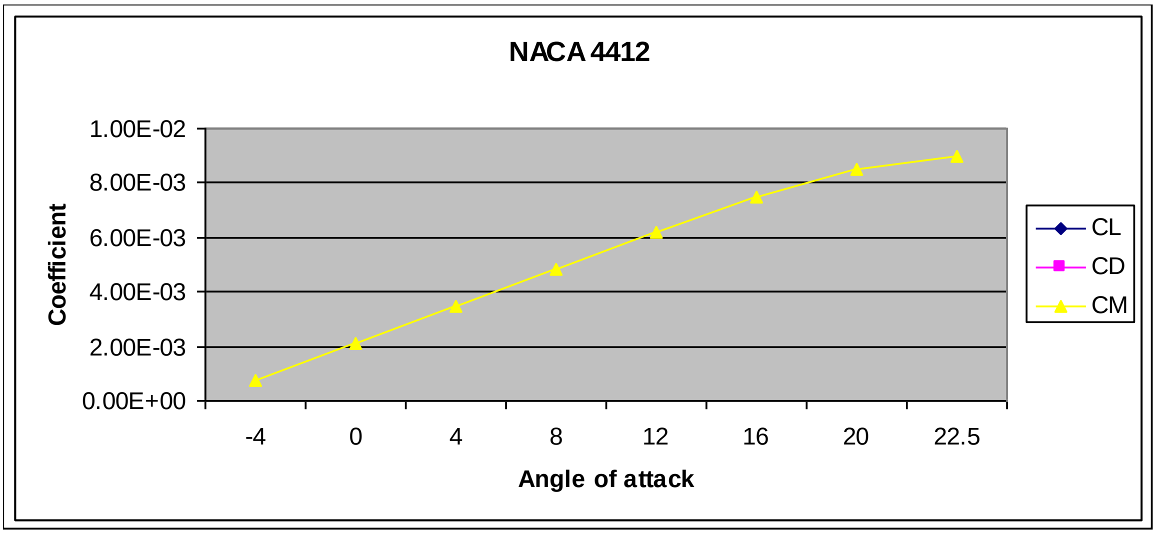

Figure 4.3.7: Pitching moment coefficient vs. angle of attack

4.4 Experimental Results by Wind Tunnel Test

The final results obtain in this project are from the experimental test by using suction type subsonic wind tunnel. Because it's only subsonic wind tunnel, the maximum velocity of this wind tunnel only can reach up to 60m/s and it has 300mm x 300 mm x 1500 mm test section. Figure 4.4.1 shows the wind tunnel that has been used for this project.

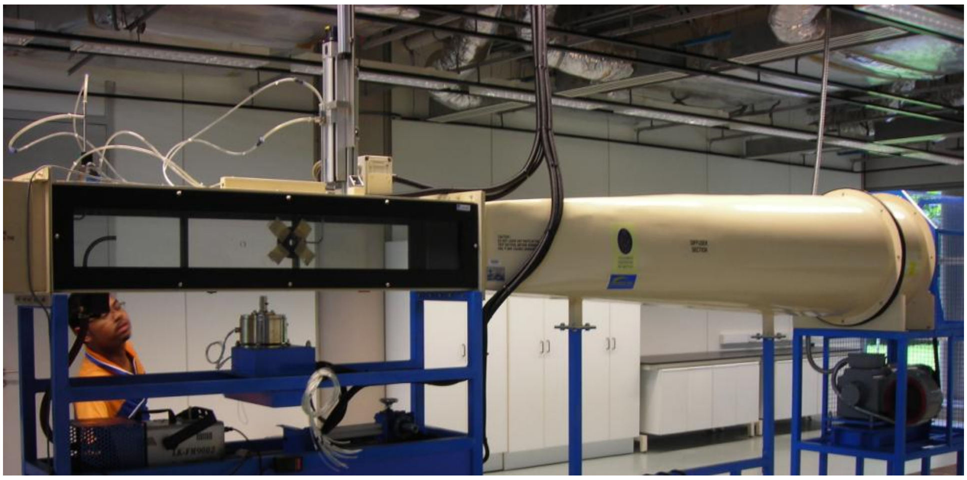

Figure 4.4.1: Subsonic wind tunnel

In the test section, the trailing edge of the airfoil was used to indicate the angle of attack of the airfoil because it has thin shape compare with the leading edge. There is Figure 4.4.2 for reference of the airfoil position during 4º angle of attack experiment.

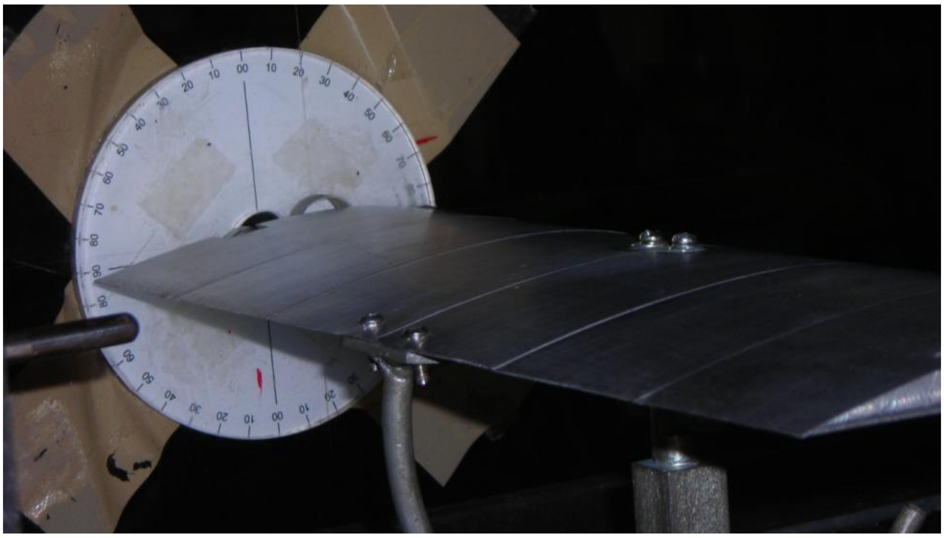

Figure 4.4.2: Airfoil in the test section for 4º angle of attack experiment

After setting airfoil into the test section, the velocity was applied by increase it slowly up until 46m/s as used in the simulation before. When the velocity reach the 46m/s, data were gathered which are forces of lift, drag and pitching moment that will be used to calculate the coefficients after that. However, the accurate result cannot be obtained because the readings of the forces keep fluctuating. The overall results for experimental was shown in Table 4.4.1 below.

Table 4.4.1: Overall coefficient of the simulation

Angle of attack	Fl (N)	Fd (N)	Pm (N.m)	Cl	Cd	Cm
-4	5.31	-0.64	-0.24	0.12979	-0.01564	0.055869
0	15.35	4.12	-0.03	0.375193	0.100703	0.006984
4	20.2	2.8	-0.21	0.49374	0.068439	0.048885
8	24.96	2.17	-0.03	0.610087	0.05304	0.006984
12	33.15	5.24	-0.06	0.810271	0.128079	0.013967
16	38.6	5.51	0.08	0.943483	0.134679	-0.01862
20	40.52	6.43	-0.02	0.990413	0.157166	0.004656
22.5	44.26	7.03	-0.08	1.081828	0.171831	0.018623

In order to interpret the data, once again Microsoft Excel becomes a critical tool to plot the graph as done in simulation previously. The coefficients lift data from the experiment quite similar with the simulation data by keeping increase with wider angle of attack shown by Figure 4.4.3. The airfoil also looks like to experience stall after reaching angle of attack of 20^{o}. On the other hand, the drag coefficient also increase, but not similar with the simulation because the data fluctuate. Somehow, the increases of drag with wider angle of attack still meet the theory discuss previously shown by Figure 4.4.4. Lastly, the pitching moment coefficients cannot take into consideration because the data fluctuate too much and the range of value too small to take into consideration, which accuracy around 0.05 shown by Figure 4.4.5.

A few errors might be occurring during the experiment due to;

- The nut to pin the airfoil into the test section has provide a little bit of roughness to the surface of the airfoil.
- The airfoil cannot join into the test section horizontally well because slip occurs between the mirror box and the airfoil.
- The data fluctuate and not show any tendency to become stable in order to take the data.

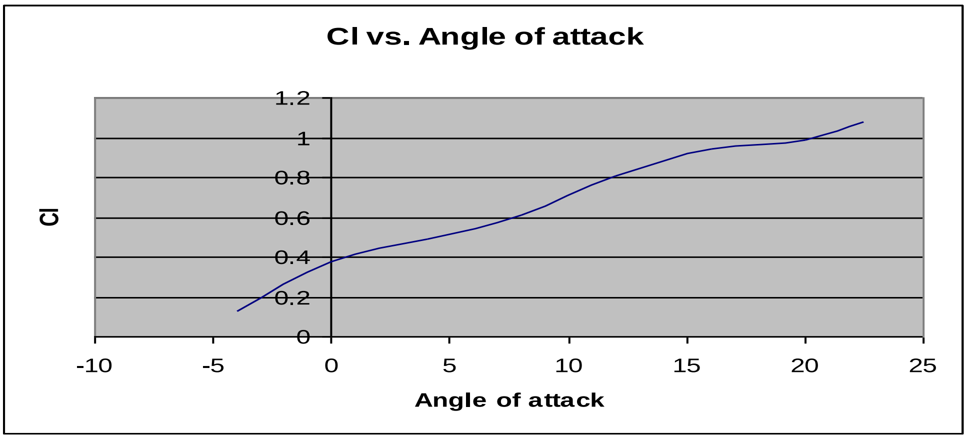

Figure 4.4.3: Lift coefficient vs. angle of attack

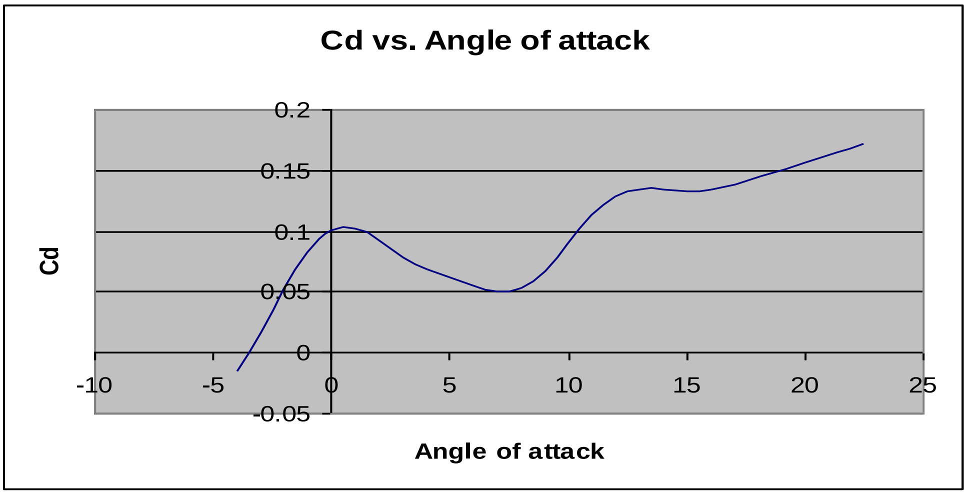

Figure 4.4.4: Drag coefficient vs. angle of attack

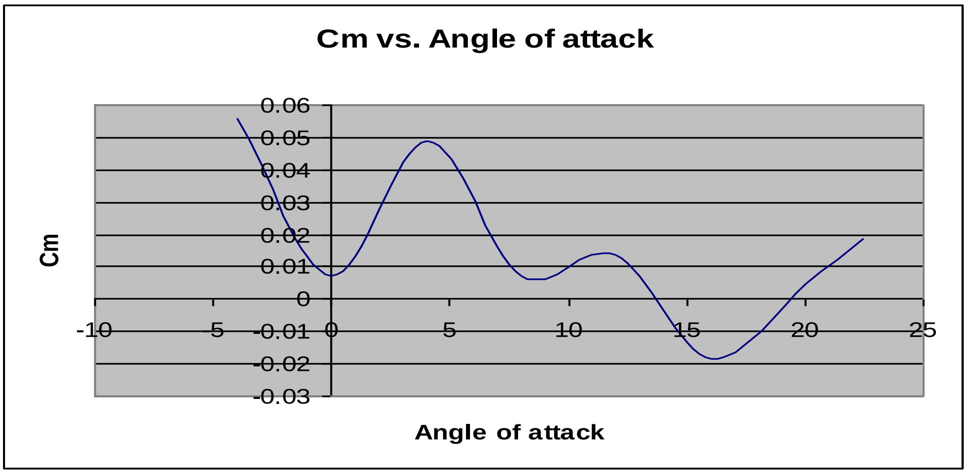

Figure 4.4.5: Pitching moment coefficient vs. angle of attack

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

At the first stage simulation by using ANSYS, the result obtained only for pressure distribution and velocity distribution for 0º and 4º angle of attack. Furthermore, it provides the coefficient of pressure, C_P by using certain equation. By considering the option available, a new model created by using GAMBIT, simulated by FLUENT, the results obtained give the data for coefficient of pressure, C_P with the results of coefficient of lift, C_L coefficient of drag, C_D, and coefficient of pitching moment, C_M. It followed by the experimental airfoil testing by using the wind tunnel and come up with the coefficient required.

As the conclusion, aerodynamics characteristics of the NACA 4412 airfoil wing type can be revealed by using FLUENT as the CFD tool and wind tunnel test. Both of the finding show that coefficient of the lift increase with wider angle of attack. Furthermore, form the simulation, it look likes the lift would be have stall after 20º. For the drag, the coefficient of the drag also increase with wider angle of attack and can be proved by both simulation and experiment also. But, for the pitching moment coefficient, major different between simulation and experiment with a few error occur have to be examine for further research before can be concluded. Furthermore, a Reynold Number value is affecting the characteristic investigation of the airfoil and this is the reason for the small differences of recent results from others for example attached in Appendix 6.

Finally, the result proved why this high lift wing uses in the jet fighter industry because it has big angle of attack before it stall. It very useful for the jet fighter to moving faster avoid the attacks because the wing section have high coefficient lift and no need to worry about stall.

5.2 Recommendations

From all the data obtained, the objective of the project should be considered success. But, in order to give more accuracy to the findings, a few methods can be implemented into both simulation and experiment. The effect of Reynolds Number also must be taken into consideration before doing some comparison between works.

For the simulation, in order to get more clear when the stall occurs, the simulation can be increase by wider the angle of attack to find the point where actually the coefficient lift stall. Although ANSYS cannot provide solution for the simulation, the pressure distribution or velocity contour results can be take into consideration, if it can provided such way to change the angle of attack.

For the experiment, the surface of the airfoil must be smoother with change the method to pin it to the test section. The slip between airfoil and mirror box also have to be removed in order to avoid vibration occur that maybe lead to decrease the fluctuation of the data and finally give more precise result, especially for coefficient of pitching moment.

REFERENCES

[1] Yunus A. Cengel and John M. Cimbala, 2006, *Fluid Mechanics: Fundamental and Applications*, Singapore, Mc Graw Hill.

[2] Andy J. Keane and Prasanth B. Nair, 2005, *Computational Approaches for Aerospace Design: The Persuit of Excellence*, England, John Wiley & Sons Ltd.

[3] R.H. Barnard and D.R. Philphot, 1995, *Aircraft Flight*,, England, Prentice Hall

[4] http://en.wikipedia.org/wiki/Wind_tunnel

[5] http://books.google.com/books?id=o6fAL6bxLBIC&pg=RA2-PA649&lpg=RA2-PA649&dq=naca+4412+airfoil+wing+section&source=web&ots=iYTAFtvBHa&sig=LatDgntiK3VMT3ektRkQrRzqlHQ

[6] John D. Anderson, Jr, 2007, *Fundamentals of Aerodynamics*, USA, Mc Graw Hill.

[7] http://en.wikipedia.org/wiki/Turbulence

[8] H H Al-Kayiem, MSc, A J Day, MA, PhD and H H Bruun, PhD, *A computer aided manufacturing procedure for experimental two-dimensional aerofoils*, Mechanical Engineering Department, M E C. Baghdad, Iraq and Department of Mechanical and Manufacturing Engineering, University of Bradford, West Yorkshire.

[9] http://www.pagendarm.de/trapp/programming/java/profiles/NACA4.html