CHAPTER 1
INTRODUCTION

1.1 BACKGROUND OF STUDY

Flow through object phenomena is very important subject in aerodynamics. This phenomenon gave more information towards fluid flow characteristics and its form formation. Analysis in two dimensional already being used to describe this phenomenon and it's enough for simple situation. Nowadays, it's already known that the flow phenomenon through an object is more complex, two dimensional analyses are no more enough to show the characteristic of the flow.

Recently, the complexity of three dimensional flows around airfoil-flat plate with the existence of gap between airfoil-flat plate tip and wall has attracted many researches. Majority of the numerical study on three dimensional flows are conducted to compressor cascade. This is due to the wide range of usage of the compressor cascade despite of it's operate at higher pressure and velocity. This study stressed on single body of airfoil-flat plate combination with gap. Through the numerical study, the Computational Fluid Dynamics (CFD) has been used to analyze the characteristics of 3-D flow of airfoil-flat plate combination with gap.

1.2 PROBLEM STATEMENT

Wind tunnel is important equipment for experimentation. Experiment on airfoil-flat plate also done in the wind tunnel. Most of the experiment that has been done is related to two dimensional flows. The effect of wall and gap is not permitted. Ideally, there is no gap. But, actually, there is gap which will disturb the accuracy of data. So, we need to study the flow around airfoil and the wall with presence of gap.

The flow on airfoil-flat plate with existence of gap between airfoil-flat plate and wall actually cannot be considered as two dimensional flows anymore. Three dimensional flows already took place for that situation. When deal with three dimensional flows, the flow that exist is actually very complex. The turbulent flow took place; contribute to unsteadiness flow and this lead to the growth and development of a horseshoe vortex. The small leakage flow through the tip gap has an effect on the loss and this unavoidable tip gap that exists complicate the three dimensional picture. The real picture of the three dimensional flow need to be knows. Right now, there’s no numerical study to know the three dimensional flow characteristic of the airfoil-flat plate combination with gap.

Besides that, it’s also necessary to know the pressure coefficient of the airfoil-flat plate, pressure distribution at the wall, pressure difference between upper and lower side of airfoil-flat plate, pressure gradient and where the saddle point form. To get more variable result, study on different type of airfoil-flat plate will help to show different result such as pressure coefficient of the airfoil-flat plate, pressure distribution at the wall, pressure difference between upper and lower side of airfoil-flat plate, pressure gradient and where the saddle point form. Using different gap clearance thickness also can contribute to variation in results.

1.3 OBJECTIVE AND SCOPE OF STUDY

1.3.1 Objective

Through numerical study on the three dimensional flow of airfoil-flat plate combination with gap, the objectives of the study are:

- To know the properties of air flow at airfoil-flat when there is existence of gap between the airfoil-flat plate tip and wall.
- To analyze the three dimensional flow characteristic that produce after simulation using CFD software.

This situation happens in the wind tunnel when there is experiment on airfoil. Through this study, the influence of the gap between airfoil-flat plate tip and wind tunnel wall can be analyzed. The complexities of the 3-D flow will then be discovering.

1.3.2 Scope of Study

For this study, airfoil-flat plate will be used. The experiment for this airfoil-flat plate actually has been done in the wind tunnel. So, the same study will be conducted but this time through the numerical study which is through Computational Fluid Dynamics (CFD) software. Computational Fluid Dynamics (CFD) software is a computational technology that enables to study the dynamics of things that flow. Through CFD, a computational model can be build to represents a system or device depends on what want to study. Then, apply the fluid flow to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena.

Through CFD simulation process, there are essentially three stages: preprocessing, solving and post processing. Each stage has its own function to complete the simulation process. Normally using this CFD software it will build and analyzing a flow model. But, in this case, CATIA is used to build up the model of the airfoil-flat plate. Then only the CFD simulation process takes place which through GAMBIT software and FLUENT software.

- *Preprocessing*
  - It includes building the airfoil-flat plate geometry model using CATIA software.
  - After that, GAMBIT software is used to creating and applying computational mesh, and then specifies the boundary types.

- *Solving*
  - The CFD solver does the flow calculations and produces the results using FLUENT software.

- Post processing
  - It involves the organization and interpretation of the predicted flow data and then production of CFD images.

Due to the problem statement which is to know the pressure coefficient of the airfoil-flat plate, pressure distribution at the wall, pressure difference between upper and lower side of airfoil-flat plate, pressure gradient and where the saddle point form, two types of airfoil-flat plate will be used for study; *symmetrical airfoil* and *nonsymmetrical airfoil*. Both types of airfoil will go the same testing using CFD and the result for both airfoils will be compared.

This numerical study will be implement by putting inlet velocity is 25 m/s while air density and viscosity are assumed constant. This simulation also conducts at atmospheric pressure. There is also variation of gap clearance thickness, d between 0.066m to 0.3m which will be carried out numerically through CFD software.

CHAPTER 2
LITERATURE REVIEW AND THEORY

2.1 LITERITURE REVIEW

The evidence of two dimensional history flows was strongly influenced three dimensional flow characteristics. Three dimensional flows happen when there is interruption. This can lead to secondary flow. There will be turbulent flow and in other way lead to the growth and development of a horseshoe vortex system.

Before this, there is already experimentally study about the three dimensional flow of the airfoil-flat plate combination with gap. Experimental study was conducted in the wind tunnel by means of measurement of static pressure on wall and airfoil which endwall and midspan are included. It was evidenced that two dimensional histories was strongly influence three dimensional flow characteristics.

Three dimensional flows characteristic involve many aspects. Research in this area consist of many analysis depend on interest and objective. Start from Kubendran [2] who research on the turbulent flow at the connection of airfoil-flat plate. They found that the quantity of turbulent flow influence by energy and the location of horseshoe vortex. It is also reported that the influence still can be feel until three times chord length at the back [3]. Measurement in friction and static pressure shows that geometry blade variation and secondary flow determine the direction and energy from friction [4]. Theory from Itoh [5] connects effect of wall and flow of the curve line with the stability of the three dimensional boundary layers. From this research, it conclude that amount of Reynold number have the lowest value at the front of leading edge. Rahman [6] reported that three dimensional flows at the interaction area of symmetrical airfoil with flat plate incline to deflect to the suction side if there is combination with gap. Increase in angle of attack cause saddle point to form further away in front of leading edge. Table 2.1 summarized of important literature review.

Table 2.1: Summary of important literature review

Year	Title / Author	Journal Summary
1986	Turbulent Flow Around a Wing-Fuselage Type Juncture [2] L. R. Kubendran	The presence of the wing affects the mean flow distribution upstream of the juncture. The oncoming fuselage boundary layer separates ahead of the wing leading edge, resulting in a vortex which rolls up and trails downstream. In the juncture, the secondary flow system transports turbulence as well as modifying the mean flow and has a large effect on the distribution of turbulent stresses; the separation vortex plays a dominant role in this process. As the downstream distance from the leading edge increases the vortex diffuses, and its core moves away from the juncture. The vortex strength and the stream wise location of the core of the vortex are affected by the slenderness ratio of the wing leading edge. In the juncture flow, there is considerable evidence of similarity between the turbulent shear stresses and the mean-flow strain rates. There is also evidence of similarity in the variations of the turbulent stress components.
1992	Three Dimensional Flow at the Junction Between a Turbine Blade and End-Wall M. Y. Jabbari, R. J. Goldstein, K. C. Marston, and E. R. G. Eckert	A visualization of the flow on the suction side and end-wall of a passage between two neighboring turbine blades is compared with mass (heat) transfer measurements on the same surfaces. Besides the horseshoe and passage vortices, there are several smaller vortices formed near the junction of blade and end-wall whose origins are discussed. The vortices detach from the end-wall and move up the blade's span. These vortices, sometimes in counter rotating pairs, are responsible for substantial local variations of heat transfer.

Year	Title / Author	Journal Summary
1993	An experimental study of a turbulent wing-body junction and wake flow J.L. Fleming, R.L. Simpson, J.E. Cowling and W.J. Devenport	Extensive measurements were conducted in an incompressible turbulent flow around the wing-body junction formed by a 3:2 semi-elliptic nose/NACA 0020 tail section and a fiat plate. Mean and fluctuating velocity measurements were performed adjacent to the wing and up to 11.56 chord lengths downstream. The appendage far wake region was subjected to an adverse pressure gradient. The authors' results show that the characteristic horseshoe vortex flow structure is elliptically shaped forming the primary component of the stream wise vorticity. The stream wise development of the flow distortions and vorticity distributions is highly dependent on the geometry-induced pressure gradients and resulting flow skewing directions. The primary goal of this research was to determine the effects of the approach boundary layer characteristics on the junction flow. To accomplish this goal, the authors' results were compared to several other junction flow data sets obtained using the same body shape. The trailing vortex leg flow structure was found to scale on T. A parameter known as the momentum deficit factor was found to correlate the observed trends in mean flow distortion magnitudes and vorticity distribution. Changes in δ/T were seen to affect the distribution of velocity, with lower ratios producing well defined local turbulence maxima. Increased thinning of the boundary layer near the appendage was also observed for small values of δ/T.

Year	Title / Author	Journal Summary
	Development of Tip Clearance Flow Downstream Of a Rotor Blade With Coolant Injection From a Tip Trench Debashis Dey & Cengiz Camci	Rotational frame velocity and pressure measurements were made downstream of a rotor blade row in a single stage turbine research rig. Measurements were taken at two axial locations to track the development of the secondary flow in the blade tip region using a five hole probe. Coolant mass flow was injected at several locations on the rotor tip to investigate the effect of coolant air on the secondary flow. The ultimate objective is to reduce losses by the introduction of high momentum air in the tip gap. Results indicate that the passage and the leakage vortices retain their structures up to at least 46% chord length downstream of the rotor trailing edge. The cooling air, which is 0.3 % of the total mass-flow, appears to be well mixed with the leakage flow and makes little difference to the flow field downstream of the rotor. It appears that a relatively large amount of cooling air ought to be injected from specific locations on the rotor tip to cause any significant change in the flow field. This study summarizes the early stages of a multi-year investigation on the fluid mechanics and heat transfer aspects of turbine tip leakage flow structures.

2.2 THEORY

For this topic, several important key points need to be clear for easier in the future. Below are the important key points:

a) Airfoil-flat plate

From the study, it focuses on the airfoil-flat plat of the wind tunnel. Airfoil is the shape of a wing or blade (rotor or turbine) or sail. An airfoil body moved through a fluid produces a force perpendicular to the motion called lift. [7] Figure 2.1 show the airfoil and its components.

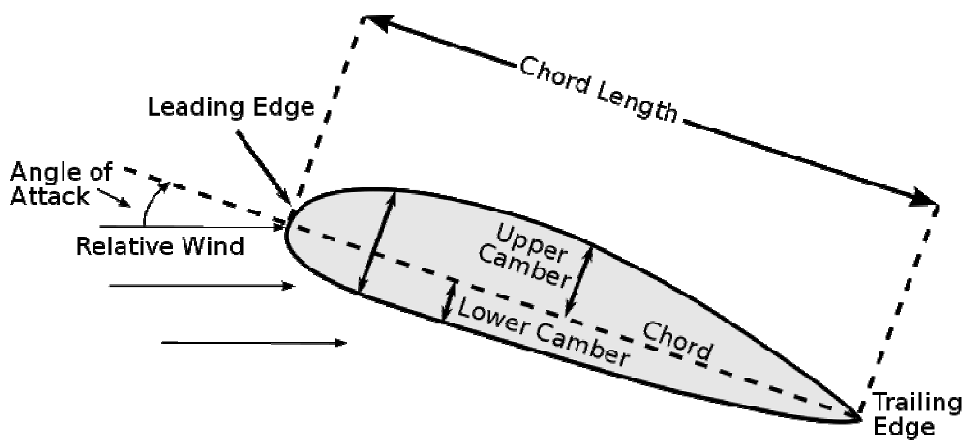


Figure 2.1: Components of the airfoil

b) Wind Tunnel

A wind tunnel is a research tool used in aerodynamic research. It is used to study the effects of air moving past solid objects.

c) Laminar flow

Occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion.

d) Turbulent flow

Occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce random eddies, vortices and other flow instabilities.

e) Pressure Coefficient

Is a dimensionless number which describes the relative pressures throughout a flow field. Every point in a fluid flow field has its own unique pressure coefficient, C_p. Pressure coefficients can be determined at critical locations around the model, and these pressure coefficients can be used with confidence to predict the fluid pressure at those critical locations around airfoil.

f) Vortex

A vortex is a spinning, often turbulent, flow of fluid. The motion of the fluid swirling rapidly around a center is called a vortex. The speed and rate of rotation of the fluid are greatest at the center and decrease with distance from the center.

Vortex is applied to gases, which have the same properties as liquids. Here, no void is created, but only an area of lower pressure, but again, a backflow causes the gas to rotate.

g) Horseshoe vortex

The horseshoe vortex model is a simplified representation of the vortex system of a wing. In this model the wing vorticity is modeled by a bound vortex of constant circulation, travelling with the wing, and two trailing vortices, therefore having a shape vaguely reminiscent of a horseshoe.

CHAPTER 3
METHODOLOGY

3.1 RESEARCH METHODOLOGY

3.1.1 Information Gathering

All the information about the three dimensional flow of airfoil-flat plate combination with gap is gathered. The information is obtained by referring to the books, journals, and search engine in the internet. Instead of gathering information about three dimensional flow of airfoil-flat plate combination with gap, the information about Computational Fluid Dynamics (CFD) software, type of flow that possible being produce and type of airfoil also gathered.

3.1.2 Type of Airfoil-Flat Plate

Through Computational Fluid Dynamics (CFD) simulation process, two types of airfoil-flat plate being used which is symmetrical airfoil and non-symmetrical airfoil. For each type, there is specific model of airfoil based on 4-digit NACA airfoil. The NACA airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA).

- Symmetrical airfoil
  - **NACA0020**

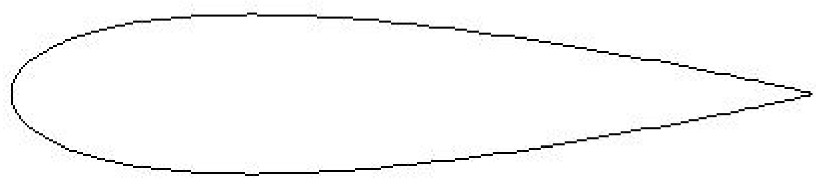


Figure 3.1: Symmetrical Airfoil

- Nonsymmetrical airfoil
  - **NACA8516**

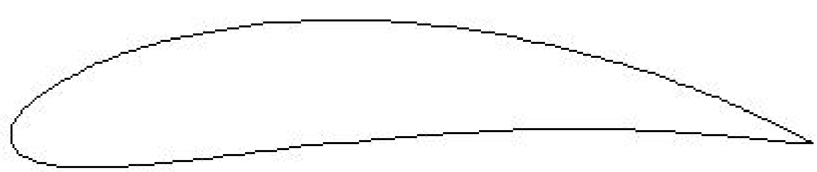


Figure 3.2: Nonsymmetrical Airfoil

3.1.3 Gap Clearance Thickness

Gap clearance thickness, d is located between the tip of airfoil-flat plate and the wall as in figure 3.3. For the gap clearance thickness, d, there is variation of thickness between 0.066m to 0.3m. Difference in gap thickness clearance also possibly can show difference in the result. Table 1 below shows the gap thickness clearance that will be used for both types of airfoils.

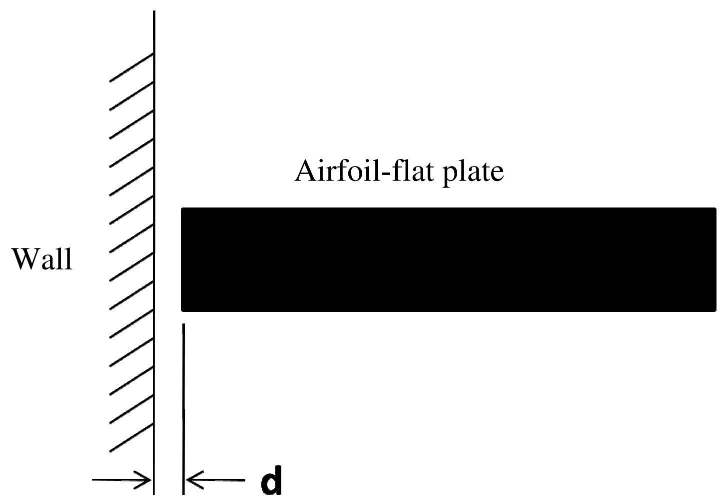


Figure 3.3: Gap clearance thickness

Table 3.1: Gap Thickness Clearance

NACA0020	NACA8516
0.066 m	0.066 m
0.120 m	0.120 m
0.180 m	0.180 m
0.240 m	0.300 m
0.300 m	

3.1.4 Dimension of the Geometry

Through CATIA software, the geometry model of airfoil-flat plate for symmetrical and non-symmetrical is created in 3-D form. Before start design the geometry, the dimension of the airfoil-flat plate and its testing area covered. From the discussion, we decide that the chord length is 100mm. Then for the testing area, length for the inlet and outlet should be 10 times longer. While for upper and lower wall, the length should be 5 times longer. So, we make a decision to draw the geometry according to the dimension as state in figure 3.4.

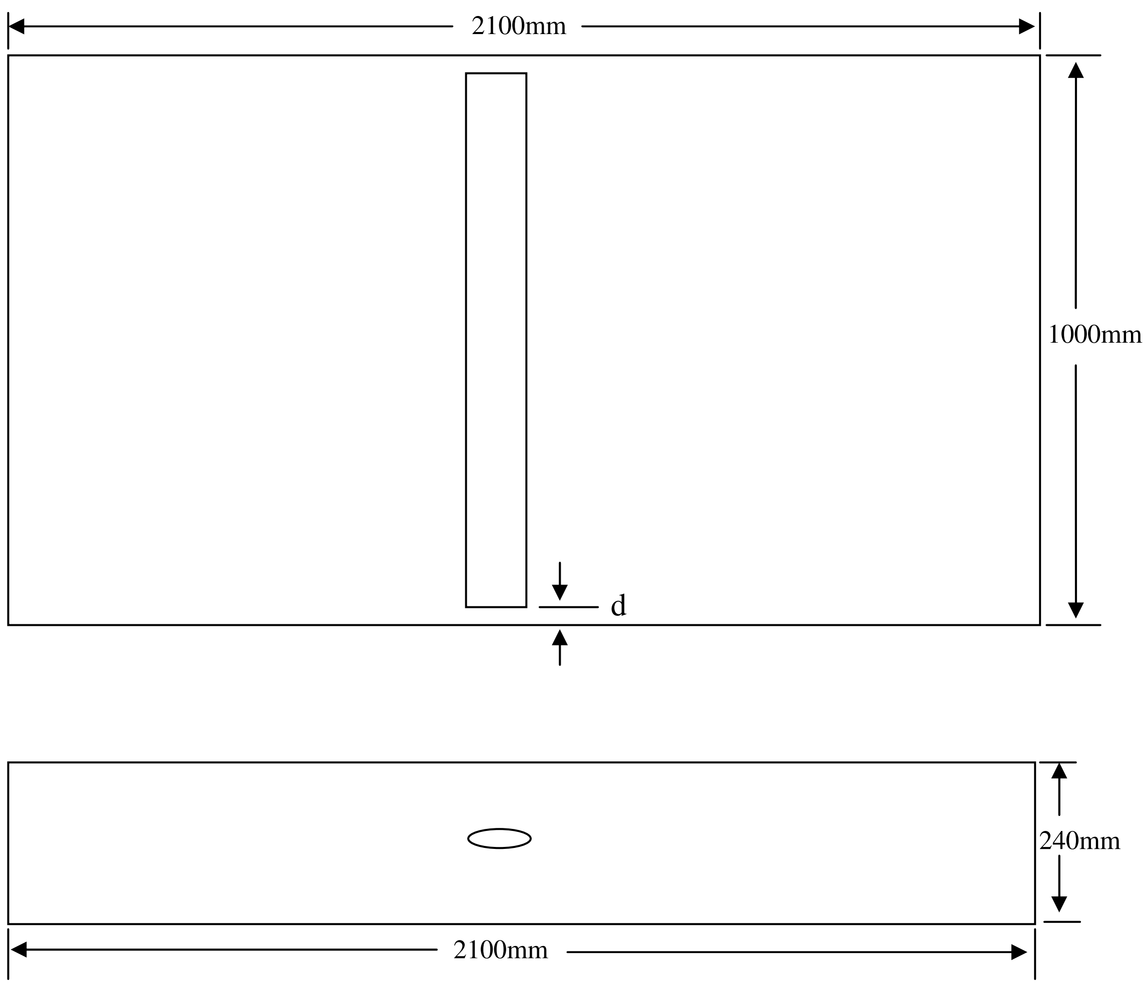


Figure 3.4: Dimension of airfoil-flat plate and its testing area

3.1.5 Computational Fluid Dynamics

Numerical study will be implementing using the CFD software. During the study, three types of software will be used, which are; CATIA software, *GAMBIT software* and *FLUENT software*. This software is used to analyze the 3-D flow of airfoil-flat plate combination with gap. Below is the procedure that used:

a) *Create airfoil-flat plate using CATIA software*

Through *CATIA software*, the geometry model of airfoil-flat plate for symmetrical and non-symmetrical is created in 3-D form. Then, the geometry model was saving into .stp file type. This type of file is only for 3-D drawing. Figure 3.5 below show the interface of *CATIA software* with the geometry of NACA0020 (symmetrical airfoil-flat plate).

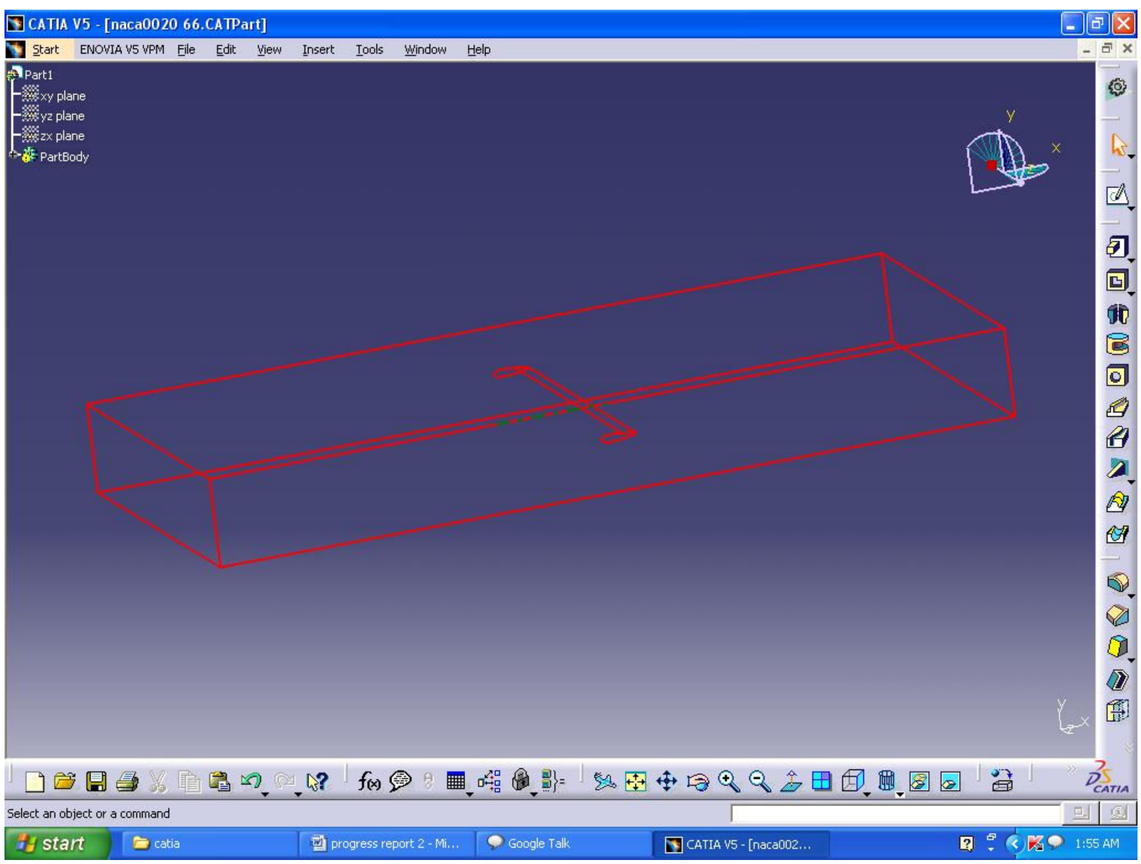


Figure 3.5: Interface of CATIA software

b) Meshing using GAMBIT software

In *GAMBIT software*, the geometrical model of the airfoil-flat plate being import from the previous work in *CATIA software*. *GAMBIT software* used to create, apply the computational mesh, and then specifies the boundary types. Four types of boundary condition being specify; wall, airfoil, velocity inlet and outlet. For meshing, several alternatives can be used through the *GAMBIT software*. This step will generate the element with the same length along the geometry. Figure 3.6 shows the interface of the *GAMBIT software* with the geometry of NACA0020 (symmetrical airfoil-flat plate).

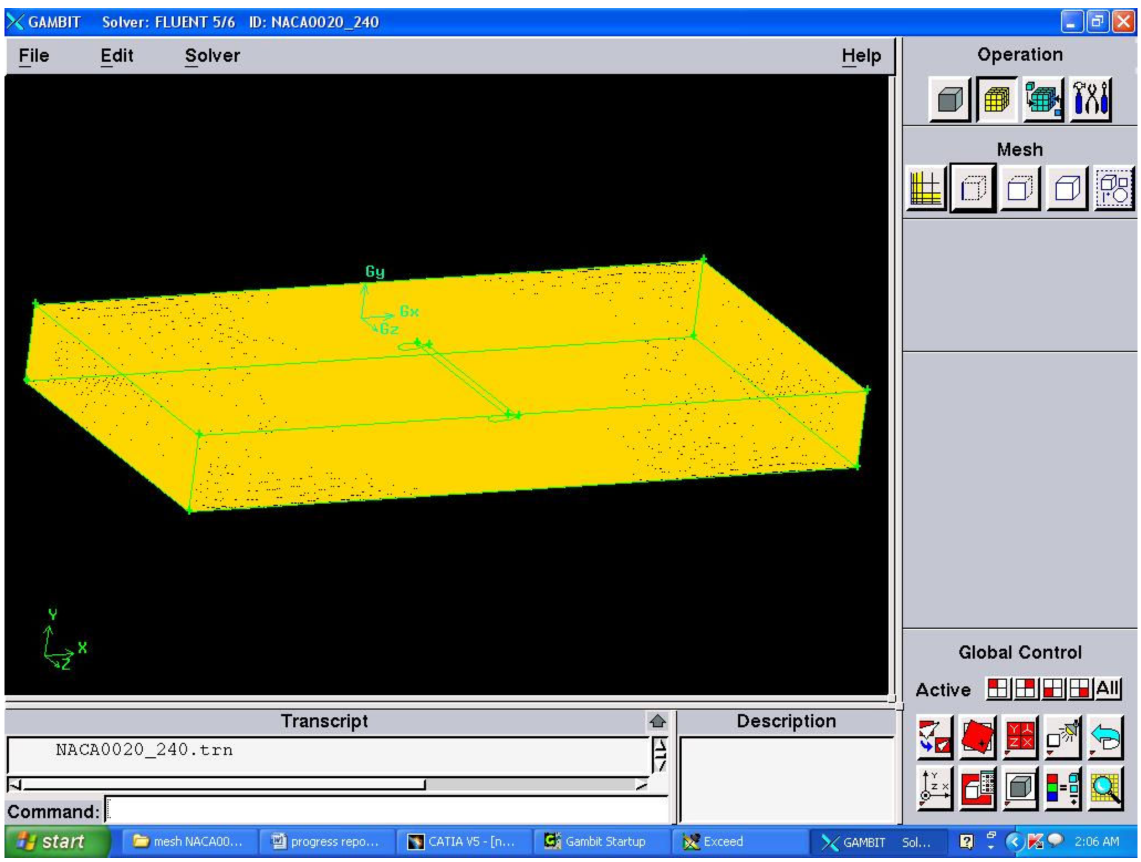


Figure 3.6: Interface of *GAMBIT software*

However, for the wall especially at airfoil-flat plate area, the mesh scale being used becomes closer. More accurate pressure distribution can be obtained at that area after simulation. Figure 3.7 shows that the scale of meshing becomes smaller at the focus area for result which is at the airfoil-flat plate area.

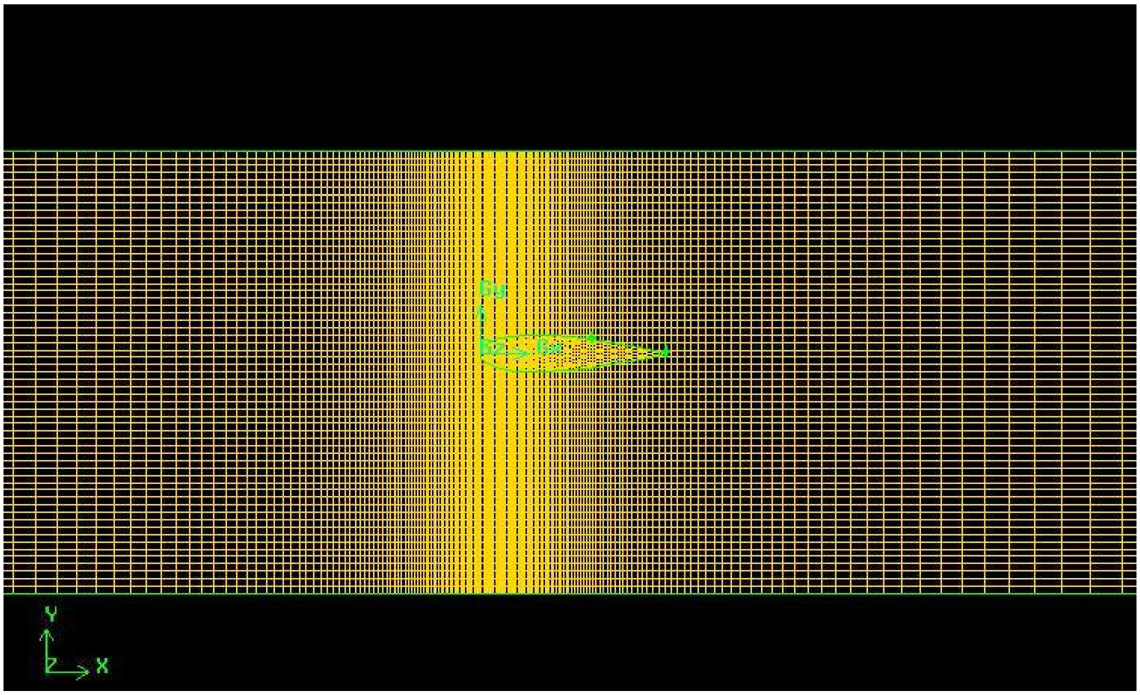


Figure 3.7: Scale of meshing becomes smaller at the airfoil-flat plate area

c) *Solution using Fluent software*

This *FLUENT software* imports the GAMBIT file. Then, it will separate into several geometry regions like wall, airfoil, velocity inlet and outflow. Operating condition should be atmospheric pressure and for boundary condition, inlet velocity, v = 25 m/s is used. For the Residual Monitors, the residual continuity is set at 0.0001. The model will be iterating until reach the expected continuity to minimize the error of simulation. Figure 3.8 shows that the iteration stops when reach the setting continuity.

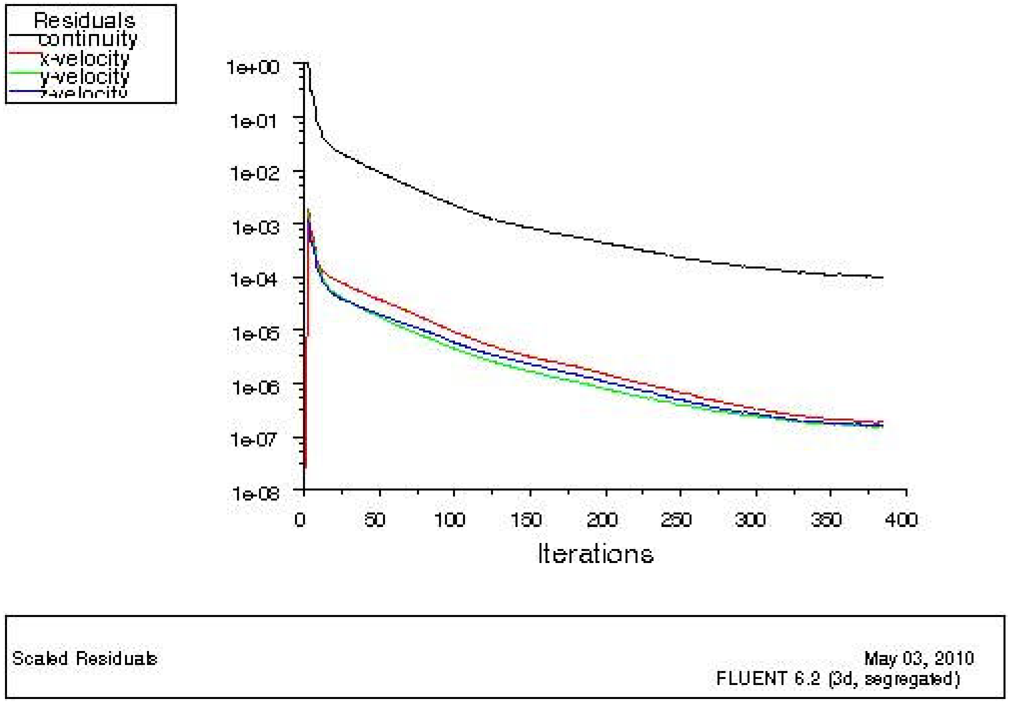


Figure 3.8: Iteration stop at Residual Continuity of 1e-4

3.2 PROJECT FLOW CHART

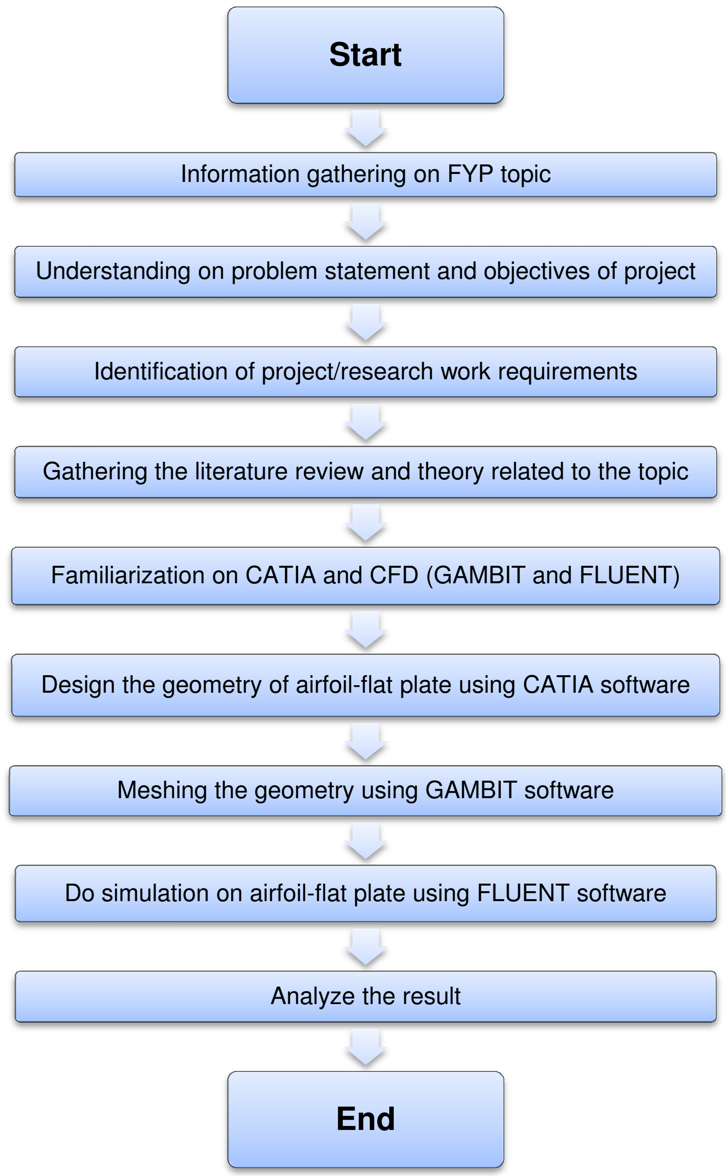


Figure 3.9: FYP Flow Chart

3.3 TOOLS

Through this numerical study, three type of software being used which are:

a. *CATIA software*

CATIA stands for Computer Aided Three-dimensional Interactive Application. CATIA is the most powerful and widely used CAD (computer aided design) software. It used to design before it could be manufactured.

b. *GAMBIT software*

The program used to design the geometry and then generate the grid or mesh for the CFD solver. Besides that it also used to specify the boundary condition. From work done in GAMBIT software; it can be import to other solver software such as FLUENT software.

c. *FLUENT software*

FLUENT software is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc.

3.4 PROJECT ACTIVITIES

A project activity that has been done is gathering information that related to three dimensional flow of airfoil-flat plate combination with gap. Besides that, CATIA software has started being used. There are 10 geometry model of airfoil-flat plate all being design through CATIA software. The next step is starting the Computational Fluid Dynamics process. Start with import the geometry of airfoil-flat plate for both symmetrical and nonsymmetrical airfoils into the GAMBIT software. The next step is using Computational Fluid Dynamics software which is FLUENT software to run the simulation and then analyze and interpret the result.

3.5 GANTT CHART

3.5.1 Final Year Project I

Table 3.1: Gantt Chart for Final Year Project I

	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Research on Topics	■													
Submission of Topic Selection Form	●													
Information gathering on FYP topic	■	■	■											
Understanding on problem statement and objectives of project			■	■										
Submission of Preliminary Report				●										
Identification of project/research work requirements					■	■	■	■						
Submission of Progress Report								●						
Seminar								■						
Gathering of literature review and theory related to the topic								■	■	■	■			
Familiarization on CFD (GAMBIT and FLUENT) by practices											■	■	■	■
Submission of Interim Report													●	
Oral Presentation														●

3.5.2 Final Year Project 2

Table 3.2: Gantt Chart for Final Year Project II

	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Design the geometry of airfoil-flat plate														
Submission of Progress Report 1				●										
Meshing through GAMBIT software														
Do simulation on airfoil-flat plate using FLUENT software														
Submission of Progress Report 2								●						
Seminar														
Analyze the result														
Poster Submission										●				
Dissertation Draft														●
Oral Presentation											During study week			
Hardbound Dissertation											7 days after presentation			

CHAPTER 4

RESULT AND DISCUSSION

Throughout the study, several variations were implemented to make better comparison of three dimensional flows characteristic. These variations then produce different results. The result was produced from the numerical simulation using *FLUENT software*. The result from the simulation is representing in form of pressure coefficient contour. From pressure coefficient contour, the properties of air flow of airfoil-flat plate include the gap can be known and then the 3 dimensional flow characteristic can be analyze. The pressure coefficient contour shows the characteristic such as pressure distribution, difference pressure between upper and down side of airfoil; the pressure gradient and the place where saddle point being form can be produce. From that, the result be analyzed and we can see the trend of the flow.

a) Type of airfoil-flat plate
   - Symmetrical (NACA0020)
   - Non-symmetrical (NACA8516)
b) Gap clearance thickness
   - o Variation between 0.066m to0.3m.

This numerical study implement by putting inlet velocity 25 m/s as initial condition while air density and viscosity are assumed constant. Table 4.1 shows the boundary condition that being implemented through the study:

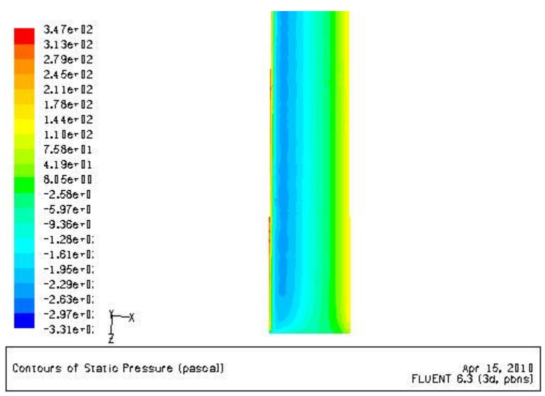
Table 4.1: Boundary Condition

Velocity, υ	25 m/s
Pressure, P	101, 325 Pa
Density, ρ	1.225 kg/m³
Temperature, T	288.16K
Kinematic Viscosity, μ	1.4507e-5 m²/s

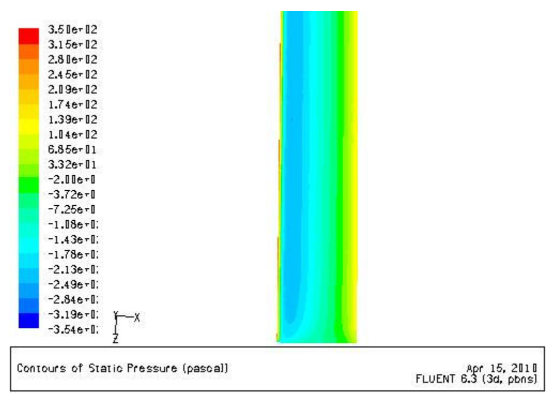
4.1 Symmetrical Airfoil-Flat Plate (NACA0020)

Under symmetrical airfoil-flat plate, varies of five gap clearance thickness is used. Each of the gap clearance thickness generates a model of airfoil-flat plate. So, that's mean there are 5 model of airfoil-flat plate being design and simulate through *FLUENT software*. From the simulation, the pressure coefficient contour is produced and then analyzes the relationship between gap clearance thickness and the pressure at the airfoil and wall area. Figure 4.1 shows the pressure coefficient contour for the upper side of airfoil-flat plate.

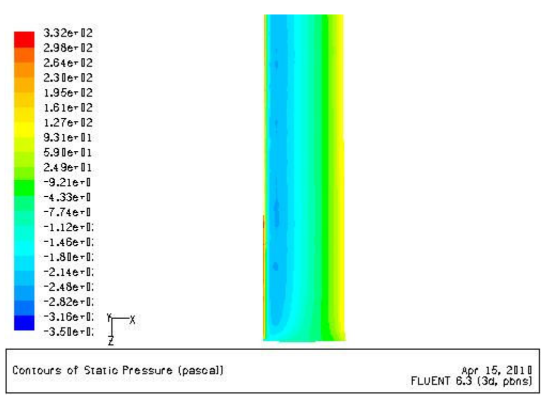
d = 0.066m



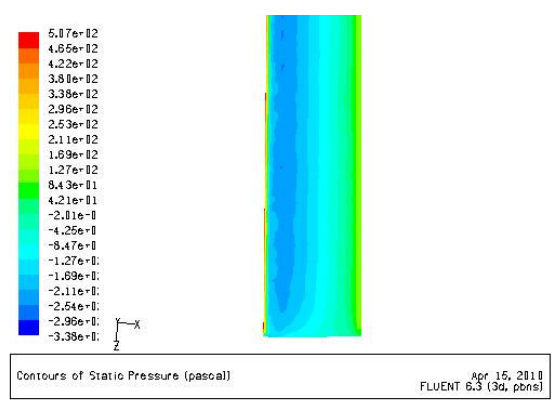
d = 0.120m



d = 0.180m



d = 0.240m



d = 0.300m

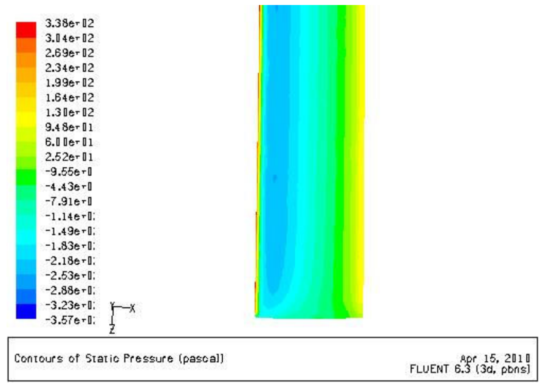
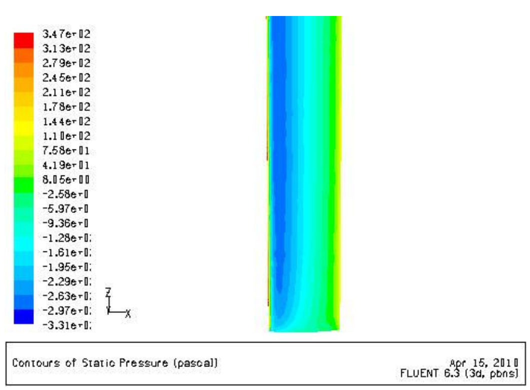


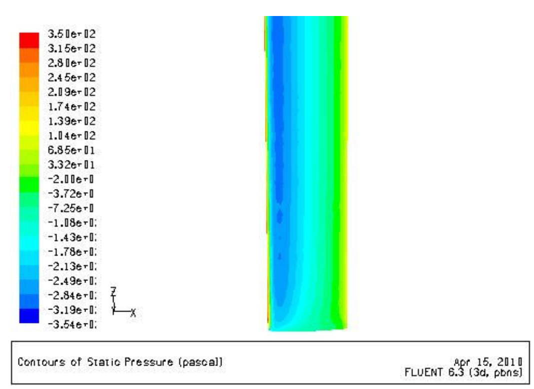
Figure 4.1: Pressure coefficient contour for the upper side of airfoil-flat plate

Based on the simulation using FLUENT software, pressure distribution can be seen along the airfoil-flat plate. At the front tip of the airfoil, from far area towards the wall to the closes area towards the wall, it only experiences a little reduction in pressure. This was shown through the colour that almost same from the middle of the body up to the tip. From the static pressure contour, it shows that the maximum pressure located exactly at the center of the front tip that marks by red or orange colour. Then, the current will speed up towards the area of the maximum thickness around 25% of chord length. This can be representing with the blue colour which indicate the lowest pressure coefficient. After that, the pressure coefficient increase until rear tip. This can be shown with colour changes from blue colour to green colour and finally yellow colour.

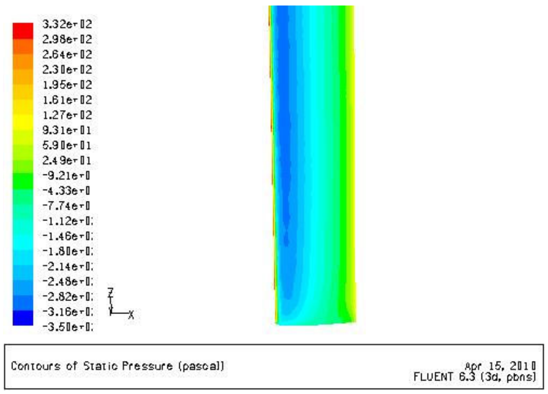
d = 0.066m



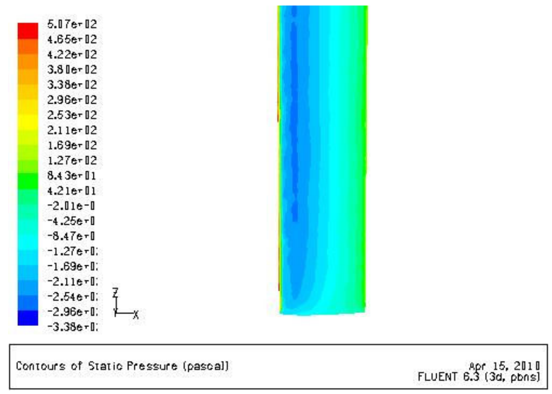
d = 0.120m



d = 0.180m



d = 0.240m



d = 0.300m

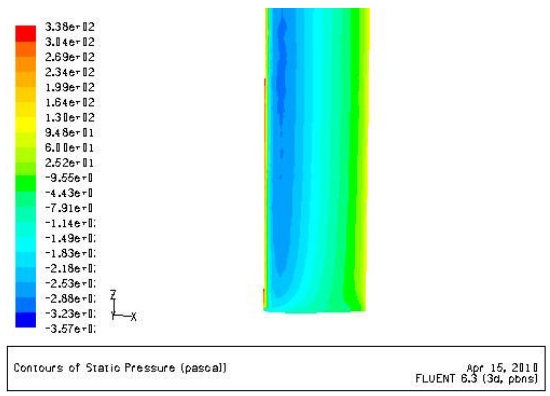
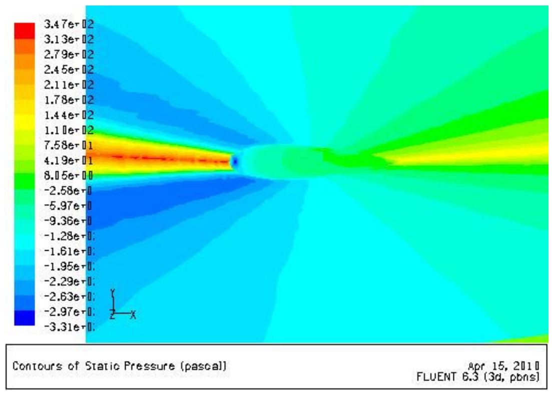


Figure 4.2: Pressure coefficient contour for the lower side of airfoil-flat plate

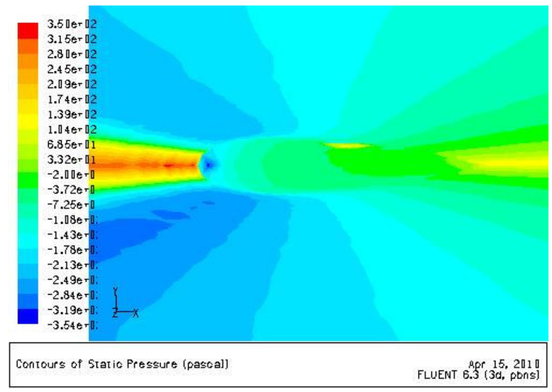
For the lower side pressure of the airfoil-flat plate, the pressure coefficient contour had the same pattern with the upper side of the airfoil-flat plate. This because of the current that going through symmetrical airfoil with the angle attack of 0º had the same current line that exactly symmetrical between upper side and lower side of the airfoil-flat plate. So, there is no pressure different between both sides. Due to no pressure distribution, vortex at the gap (between airfoil-flat plate tip and wall) will not occur. The current that enter the gap only speeded up until rear tip of the airfoil-flat plate due to entering the smaller area.

Besides that, the pressure distribution on the wall was also captured for analysis. Figure 4.3 shows the pressure coefficient contour on the wall.

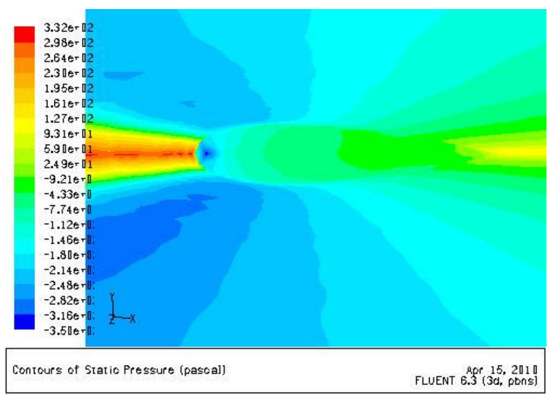
d = 0.066m



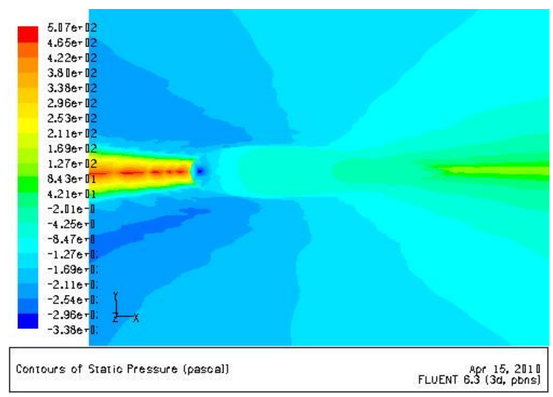
d = 0.120m



d = 0.180m



d = 0.240m



d = 0.300m

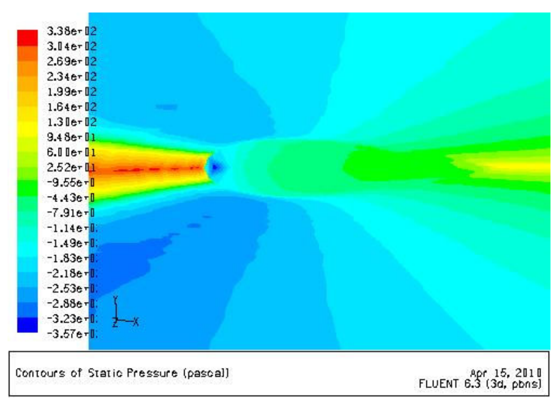


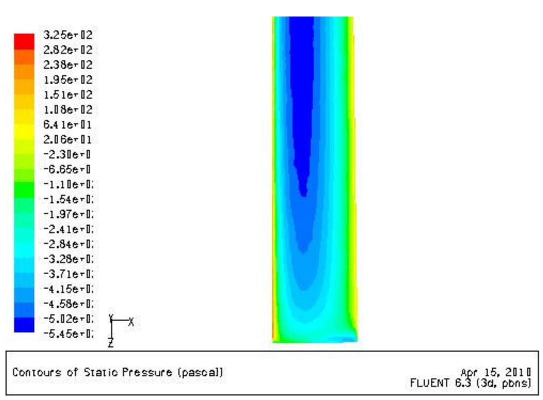
Figure 4.3: Pressure coefficient contour on the wall

From the pressure coefficient contour on the wall, it shows that the saddle point was formed exact to the chord line in front of the airfoil. So, it also concludes that the pressure distribution of the symmetrical airfoil-flat plate was same between upper side and lower side even at the wall. The saddle point was mark with blue colour at the front tip of the airfoil-flat plate and then spread from to the rear tip. We can see the change of colour from dark blue to the bright blue. That shows the effect of separation with small pressure coefficient characteristics. But, when reaching the rear tip, the colour becomes green and then yellow. It happens due to the influence of tip gap that gave the effect of suction. The effect of suction happens when current going through the gap, the current will speed up and this causes the pressure drop at the gap area. So, the current from higher pressure area tend to suck into the gap. That's why the area near to the gap has higher pressure and keeps increasing when approaching the rear tip. When the gap clearance thickness is increasing, in this case from 0.066m to 0.3m, the current speed will be decrease. The pressure drop will also be decrease. So, less current will be suck into the gap. This shows that when the gap clearance thickness increase, the pressure distribution effected by suction also be decrease.

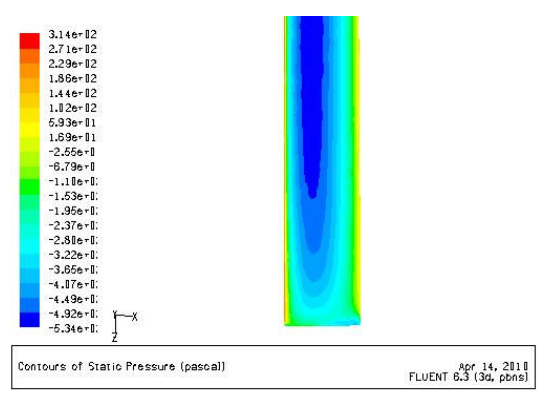
4.2 Non-Symmetrical Airfoil-Flat Plate (NACA8516)

Under non-symmetrical airfoil-flat plate, varies of four gap clearance thickness is used. Each of the gap clearance thickness also generates a model of airfoil-flat plate. So, that's mean there are 4 model of airfoil-flat plate being design and simulate through *FLUENT software*. From the simulation, the pressure coefficient contour is produced and then analyzes the relationship between gap clearance thickness and the pressure at the airfoil and wall area.

d = 0.066m



d = 0.120m



d = 0.180m d = 0.300m

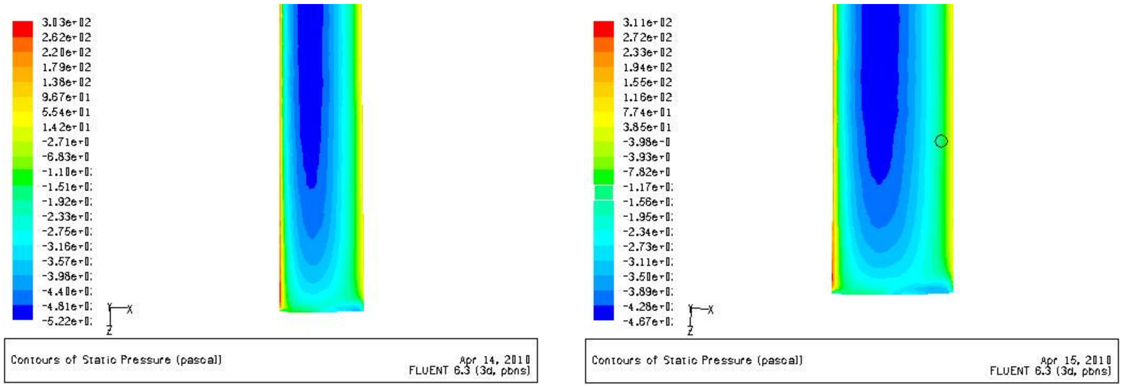


Figure 4.4: Pressure coefficient contour for the upper side of airfoil-flat plate

Unlike symmetrical airfoil-flat plate, change in lower side contour cause the pressure gradient return bigger until there is pressure difference even at the angle of attack 0º. Figure 4.4 shows the pressure coefficient contour for the upper side of airfoil-flat plate. From the pressure coefficient contour of the upper side of the non-symmetrical airfoil-flat plate, it same with the symmetrical airfoil-flat plate at angle of attack 0º. At the front tip of the airfoil, from far area towards the wall to the closes area towards the wall, it only experiences a little reduction in pressure. To the tip area (around 25% chord), the dark blue colour deflect behind was replace by light blue and this shows that there are rise in the pressure compared to the area closer to the middle of the body. However if being observed further, green colour seen more dominated down to the rear tip. This means higher pressure coefficient keeps dominate until the end of rear tip. This time we can see that more effect towards the gap between the airfoil-flat plate and wall. Due to the effect of the interaction with the wall that results in shortage of momentum until it trip towards the area of maximum thickness.

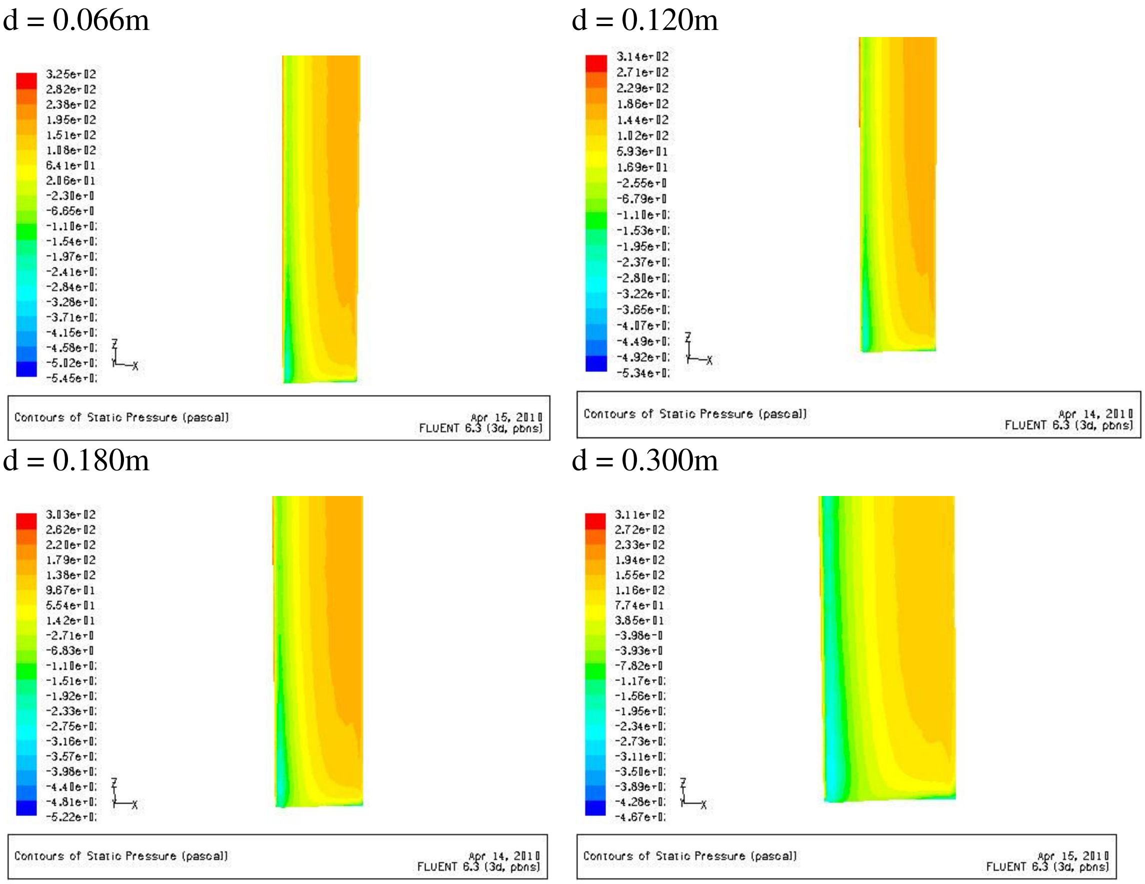
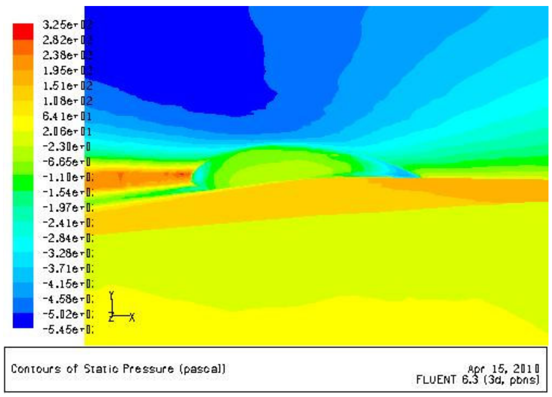


Figure 4.5: Pressure coefficient contour for the lower side of airfoil-flat plate

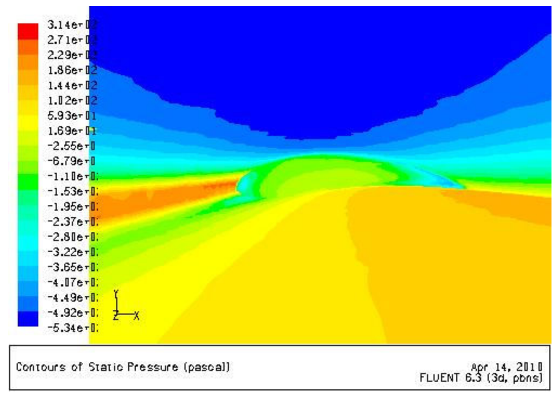
At the lower side of the airfoil-flat plate, the pressure coefficient contour is differing from the upper side of the airfoil-flat plate. Pressure coefficient contour for the lower side of airfoil-flat plate shown in the figure 4.5. At the front tip, there is orange colour. But it start trail towards the chord length, it start to change into green colour. That means there is reduction in pressure. But when it continue trail the chord the colour start to change into yellow colour and then slowly change into orange colour. That means the pressure keep increase after about 25% of chord length. For the non-symmetrical area, at the lower side of the airfoil, the current slow down, so the pressure will increase. That’s why if we compare the colour trend, for the upper side, blue colour dominates most of the area. But for the lower side, the yellow and orange colour dominates most of the area. The colour trend also shows that lower side poses higher pressure compared to the upper side.

Difference of pressure distribution causes the emergence of tip gap vortex especially near the rear tip towards the wall. At that area, the current that through the lower side got the pressure on the upper side lower than the pressure on the lower area at the rear tip. The distributions of the colour at the figure 4.5 were seen greener to the area of the rear tip than in figure 4.4.

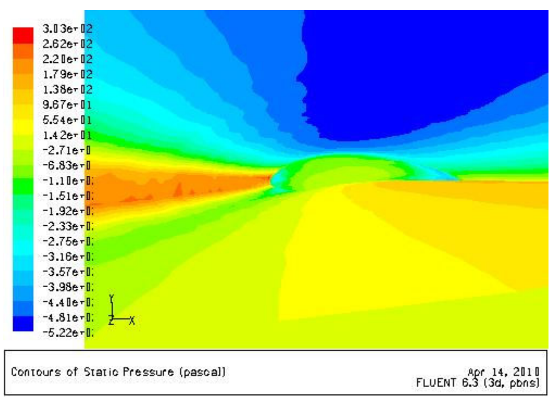
d = 0.066m



d = 0.120m



d = 0.180m



d = 0.300m

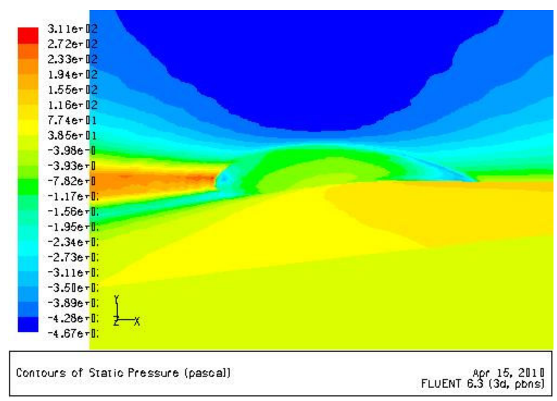


Figure 4.6: Pressure coefficient contour on the wall

Figure 4.6 shows the pressure distribution on the wall, seen that the saddle point was form exactly at the front tip of chord line, but slightly down. This shows that the difference of pressure between the upper side and lower side had the influence down to the area of whichever the front tip have the same geometry form between upper and lower side and approximately 25% chord length. Therefore the current in front of the front tip under chord line experienced bigger pressure gradient return and more often lost energy resulting from the effect of the gap.

The saddle point was marked by at the beginning with blue colour emergence. Due to pressure difference, the blue colour at the upper side of airfoil-flat plate that spread towards chord showed the influence of 3 dimensional separation were bigger at that area. The influence of 3 dimensional separation exist until the current left rear tip of the airfoil-flat plate was marked with the existence of different distribution colour even though both side experienced energy increase.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

As a conclusion, the result of airfoil-flat plate pressure coefficient contour for NACA0020 (symmetrical) and NACA8516 (non-symmetrical) are different. For NACA0020 (symmetrical), at angle of attack 0°, the pressure coefficient contour that produced for upper and lower side of the airfoil-flat plate is same. That's mean there is no pressure different between upper and lower side of airfoil and the pressure gradient is not exist. Due to no pressure different, the saddle point was formed exactly in front of the airfoil-flat plate chord line. The influences of gap cause the effect of suction which can be seen at the tip of airfoil-flat plate near the wall. Pressure distribution at that area differs from other area pressure distribution.

However, for NACA8516 (non-symmetrical), even at angle of attack 0°, the different of pressure coefficient contour between upper and lower side of airfoil-flat plate can be seen clearly. From the pressure coefficient contour, we can see blue colour dominate upper side of airfoil-flat plate while for the lower side, yellow and orange colour dominate. These phenomena definitely cause high pressure gradient. For NACA8516, the influence of gap is clearer than NACA0020 which cause the emergence of tip gap vortex especially near the rear tip towards the wall. The saddle point was form exactly at the front tip of chord line, but slightly down.

Even though there are several between symmetrical and non-symmetrical airfoil-flat plate, there is still similarity in certain aspect for both. Both airfoil-flat plates pose lowest pressure at the highest thickness of airfoil-flat plate which is about 25% of chord length. After that, the pressure keeps increasing until the rear tip.

5.2 RECOMMENDATION

Recommendation should be made to improve the result and minimize the error. So, there are some recommendation needs to be made to improve and make variation on result. Recommendations are as below:

- Variation angle of attack

For this study, only angle of attack 0° being used. If more angle of attack can be use, that's mean more pressure coefficient contour can be produced. So, differences between angle of attack can analyze.

- Set lower continuity for iteration

When lower continuity is set, that's mean more iteration needed to complete the simulation. So, it can minimize error and better for the result.

- Variation in velocity

Several velocities can be set to get different pressure coefficient contour. This is due to connection between velocity and pressure. The current will speed up when higher velocity is used and this will reflect the pressure distribution.

- Different airfoil-flat plate thickness

Different thickness of airfoil-flat plate can produce different pressure coefficient contour. This can make variation in result and better for analysis and discussion.

REFERENCES

1. Gunawan Nugroho & Herman Sasongko, Studi Numerik dan Eksperimental Aliran 3-D pada Kombinasi Airfoil/Plat datar dengan Variasi Permukaan Bawah dan Pengaruh Celah, Jurnal Teknik Mesin, Oktober, Vol. 7, No. 2, 2005.
2. Kubendran, L.R., Turbulent Flow Around a Wing/Fuselage-Type Juncture, AIAA Journal, September, Vol. 24 No.9. 1986.
3. Merati, P. & Mc Mahon, H.M. & Yoo, K.M., Experimental Investigation of a Turbulent Flow in the Vicinity of an Appendage Mounted on a Flat Plate, Journal of Fluids Engineering, December, Vol. 113/635. 1991.
4. Abdulla,A.K. & Bhargava,R.K. & Raj,R., An Experimtal Study of Local Wall Shear Stress, Surface Static Pressure, and Flow Visualization Upstream, Alongside, and Downstream of a Blade Endwall Corner, Journal of Fluid Engineering, Vol. 113/626, 1991.
5. Itoh,I., Effects of Wall and Streamline Curvatures on Instability of 3-D Boundary Layers, AIAA Journal, September, Vol. 30 No.5. 1994.
6. Rahman,A., Studi Eksperimental Aliran TigaDimensi pada Daerah Interaksi Airfoil Simetrisdan Pelat Datar dengan Pengaruh Clearancedan Perubahan Angle of Attack, Tugas Akhir Jurusan Teknik Mesin, FTI-ITS. 2003.
7. M. Y. Jabbari, R. J. Goldstein, K. C. Marston, and E. R. G. Eckert, Three Dimensional Flow at the Junction Between a Turbine Blade and End-Wall, Wfirme- und Stofffibertragung 27, 1992.
8. J.L. Fleming, R.L. Simpson, J.E. Cowling and W.J. Devenport, An Experimental Study of a Turbulent Wing-Body Junction and Wake Flow, Experiments in Fluids 14, 366-378, 1993.
9. Debashis Dey & Cengiz Camci, Development of Tip Clearance Flow Downstream Of a Rotor Blade With Coolant Injection From a Tip Trench
10. Yunus A. Cengel & John M. Cimbala (2006). Fluid Mechanics: Fundamental and Applications. Mc Graw Hill.
11. http://en.wikipedia.org/wiki/Computational_fluid_dynamics
12. http://en.wikipedia.org/wiki/Wind_tunnel