



**Faculty of Mechanical Engineering**

**NUMERICAL SIMULATION OF DEFOGGING ON AUTOMOTIVE  
WINDSHIELD**

**Faris bin Azman**

**Master of Mechanical Engineering (Automotive)**

**2017**

**NUMERICAL SIMULATION OF DEFOGGING ON AUTOMOTIVE  
WINDSHIELD**

**FARIS BIN AZMAN**

**A report submitted  
in fulfillment of the requirements for the degree of Master of  
Mechanical Engineering (Automotive)**

**Faculty of Mechanical Engineering**

**UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

**2017**

## DECLARATION

I declare that this report entitled “Numerical Simulation of Defogging on Automotive Windshield” is the result of my own research except as cited in the references. The report has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :   
Name : FARIS BIN AZMAN  
Date : 10 MARCH 2017

## APPROVAL

I hereby declare that I have read this report and in my opinion this report is sufficient in terms of scope and quality as a partial fulfillment of Master of Mechanical Engineering (Automotive).

Signature



Supervisor Name

: FUDHAIL BIN ABDUL MUNIR, DR

Date

: 10-03-2017.

## **DEDICATION**

To my beloved father and mother

## ABSTRACT

Defogging is a process to eliminate fog or water vapour from a surface such as windshield. It is crucial to fundamentally understand the fogging and defogging phenomena in order to reduce potential vehicle accidents due to visibility problems. In this study, the numerical simulation of the defogging phenomenon on the automotive windshield is presented. ANSYS-Fluent is used as the software tool for simulation where the defogging phenomenon on the windshield is investigated and the data such as velocity distribution in the model and water layer thickness and temperature distribution of the water layer film on the windshield are determined. The simulation is conducted on three cases related to relative humidity and another three cases related to the inlet size where the best defogging phenomenon on the windshield can be observed at the case with lowest relative humidity value of 54% and with the largest inlet area of 0.0225 m<sup>2</sup>.

## ABSTRAK

*Nyahnkabus merupakan satu proses untuk menghilangkan kabus atau wap air dari permukaan seperti cermin depan. Ini amat penting bagi memahami asas fenomena pengkabusan dan nyahnkabus dalam usaha untuk mengurangkan potensi kemalangan melibatkan kenderaan kerana masalah penglihatan. Simulasi berangka fenomena nyahnkabus pada cermin depan automotif dijelaskan dalam kajian ini. ANSYS-Fluent digunakan sebagai alat perisian untuk simulasi di mana fenomena nyahnkabus di cermin depan dikaji dan data seperti taburan halaju dalam model dan ketebalan lapisan air dan taburan suhu di filem lapisan air pada cermin depan ditentukan. Simulasi dijalankan ke atas tiga kes berkaitan dengan kelembapan relatif dan tiga kes lain berakitan dengan saiz salur masuk di mana fenomena nyahnkabus yang terbaik boleh dilihat pada kes dengan kelembapan relatif yang terendah pada nilai 54% dan dengan kawasan salur masuk yang terbesar pada nilai  $0.0225 \text{ m}^2$ .*

## ACKNOWLEDGEMENTS

First of all, praise to Allah for all of the good things, health and strength throughout the semester for preparing and finishing this “Projek Sarjana” or “Master Project” thesis without any problems. I would like to take this opportunity to express my sincere acknowledgement and big thanks to my supervisor Dr. Fudhail bin Abdul Munir from the Faculty of Mechanical Engineering (FKM), Universiti Teknikal Malaysia Melaka (UTeM) for the crucial support, supervision, encouragement and advice towards the completion of this thesis.

Besides that, I would like to give my thanks to all of committee of “Master Project” from FKM and Centre of Graduate Studies (CGS) for giving me this opportunity to study and research on the project entitled “Numerical Simulation of Defogging on Automotive Windshield” and also assist in the plan and guideline for the project and thesis.

Last but not least, special thanks to all my family members, peers from Master in Mechanical Engineering Automotive (MMCV) class and colleagues for the advice, knowledge, guidance, encouragement and moral support in a lot of ways that help me to complete this thesis. Finally, thank you to everyone who had been involved to the essential parts of realization of this project.

## TABLE OF CONTENTS

	PAGE
DECLARATION	
DEDICATION	
ABSTRACT	i
ABSTRAK	ii
ACKNOWLEDGEMENTS	iii
TABLE OF CONTENTS	iv
LIST OF TABLES	vi
LIST OF FIGURES	vii
LIST OF APPENDICES	ix
LIST OF SYMBOLS	x
LIST OF ABBREVIATIONS	xi
CHAPTER	
1. INTRODUCTION	1
1.1 Background	1
1.2 Problem Statement	2
1.3 Objectives	2
1.4 Scope	3
2. LITERATURE REVIEW	4
2.1 Fogging	4
2.2 Condensation	4
2.3 Defogging	6
2.4 Convection	8
2.5 Relative Humidity	16
2.6 Software	18
3. METHODOLOGY	21
3.1 ANSYS Fluent	21
3.2 Pre-Processing	22
3.2.1 Model Drawing	22
3.2.2 Model Meshing	23
3.3 Solver Setting	25
3.3.1 General Setup	25
3.3.2 Models Setup	27
3.3.3 Materials Setup	31
3.3.4 Cell Zone Conditions Setup	33
3.3.5 Boundary Conditions Setup	33
3.3.6 Solution Methods Setup	36
3.3.7 Solution Initialization Setup	37
3.3.8 Run Calculation Setup	39
3.4 Post-Processing	40
3.5 Relative Humidity Cases	41
3.6 Inlet Size Cases	41

<b>4.</b>	<b>RESULT AND DISCUSSION</b>	<b>42</b>
4.1	Relative Humidity	42
4.1.1	Velocity Distribution	42
4.1.2	Water Layer Thickness	44
4.1.3	Temperature Distribution	46
4.2	Inlet Size	48
4.4.1	Velocity Distribution	48
4.4.2	Water Layer Thickness	50
4.4.3	Temperature Distribution	52
<b>5.</b>	<b>CONCLUSION AND RECOMMENDATIONS FOR FUTURE RESEARCH</b>	<b>54</b>
	<b>REFERENCES</b>	<b>55</b>
	<b>APPENDICES</b>	<b>62</b>

**LIST OF TABLES**

<b>TABLE</b>	<b>TITLE</b>	<b>PAGE</b>
3.1	Properties of materials	32

## LIST OF FIGURES

FIGURE	TITLE	PAGE
1.1	Fogging on automotive windshield (Hembree, 2016)	1
2.1	Condensation types (a) Film condensation (b) Dropwise condensation (Faghri and Zhang, n.d.)	5
2.2	Thermography setup (Burch et al., 1993)	6
2.3	Boundary layer of velocity progression on flat surface (Scienceofdoom, 2011)	8
2.4	Laminar, transitional and turbulent flows by Reynold experiment (Chai et al., 2015)	11
2.5	Natural convection heat transfer from a hot object (Bahrami, n.d.)	12
2.6	Forced convection heat transfer via fan (TutorVista, 2016)	13
2.7	Velocity boundary layer of an (a) External fluid flow (b) Internal fluid flow (Klein, n.d.)	13
2.8	External fluid flow (a) Flow Parallel to the plate (b) Flow normal to the plate (Hardy, 2016)	14
2.9	Velocity and thermal boundary layers (Gaikwad, 2015)	16
2.10	Average relative humidity in Melaka (WeatherSpark, n.d.)	18
2.11	ANSYS Fluent version 17.1 software	19
2.12	ANSYS Fluent version 17.1 interface	20
3.1	Flow chart	21
3.2	Geometry of the model	22
3.3	Design of the model using CATIA	23
3.4	Meshed model	24
3.5	Setting the solver as three dimensional double precision	25
3.6	Report quality and check of the meshed model	27
3.7	Enabling energy equation	27
3.8	Viscous model settings	28

3.9	Species model settings	29
3.10	Eulerian wall film settings (a) Model options and setup (b) Solution method and control	30
3.11	Material settings (a) Mixture type material (b) Fluid type material (c) Solid type material	31
3.12	Windshield zone settings	33
3.13	Boundary conditions of (a) Velocity inlet settings (b) Pressure outlet settings (c) Wall settings	35
3.14	Solution methods settings	37
3.15	Solution initialization settings	38
3.16	Run calculation settings	39
3.17	Post-processed result observation area at (a) Cross section at inlet in XY plane (b) Cross section at the water layer film	40
4.1	Velocity distribution contours for relative humidity cases of (a) 54% (b) 73% (c) 91%	43
4.2	Water layer thickness contours for relative humidity cases of (a) 54% (b) 73% (c) 91%	45
4.3	Temperature distribution contours for relative humidity cases of (a) 54% (b) 73% (c) 91%	47
4.4	Velocity distribution contours for inlet area cases of (a) $0.0025 \text{ m}^2$ (b) $0.01 \text{ m}^2$ (c) $0.0225 \text{ m}^2$	49
4.5	Water layer thickness contours for inlet area cases of (a) $0.0025 \text{ m}^2$ (b) $0.01 \text{ m}^2$ (c) $0.0225 \text{ m}^2$	51
4.6	Temperature distribution contours for inlet area cases of (a) $0.0025 \text{ m}^2$ (b) $0.01 \text{ m}^2$ (c) $0.0225 \text{ m}^2$	53

**LIST OF APPENDICES**

<b>APPENDIX</b>	<b>TITLE</b>	<b>PAGE</b>
A	Gantt chart	62
B	User-defined Functions	63

## LIST OF SYMBOLS

$a$	-	Width of the cross section
$A_c$	-	Cross-sectional area
$A_N$	-	Normal projected area
$A_s$	-	Heat transfer surface area
$b$	-	Height of the cross section
$C_D$	-	Drag coefficient
$c_p$	-	Specific heat at constant pressure
$D$	-	Drag force
$D_h$	-	Hydraulic meter
$h$	-	Convection heat transfer coefficient
$k$	-	Fluid thermal conductivity
$l$	-	Length of the geometry
$L$	-	Fluid layer thickness
$Nu$	-	Nusselt number
$P$	-	Limited pressure of the water vapour that definitely existing in air-water mixture at the given temperature
$P_s$	-	Saturation pressure of the water at the given temperature of mixture
$P_w$	-	Wetted perimeter
$Pr$	-	Prandtl number
$\dot{Q}_s$	-	Heat transfer rate of the surface
$Re$	-	Reynolds number
$RH\%$	-	Relative humidity percentage
$T_s$	-	Temperature of the surface
$T_\infty$	-	Temperature of the free stream fluid
$V, v$	-	Velocity of the fluid
$\rho$	-	Density of the fluid
$\mu$	-	Dynamic viscosity of the fluid

## LIST OF ABBREVIATIONS

2D	-	Two Dimensional
3D	-	Three Dimensional
BEV	-	Battery Electric Vehicle
CAD	-	Computer-Aided Drawing
CATIA	-	Computer Aided Three-dimensional Interactive Application
CD-adapco	-	Computational Dynamics-Analysis & Design Application Company Ltd
CFD	-	Computational Fluid Dynamics
CGS	-	Centre for Graduate Studies
CPU	-	Central Processing Unit
EFW	-	Eulerian Wall Film
FKM	-	Faculty of Mechanical Engineering
HEV	-	Hybrid Electric Vehicle
HVAC	-	Heating, Ventilation and Air Conditioning
ICEM-CFD		Integrated Computer-aided Engineering and Manufacturing Computational Fluid Dynamics
MMCV	-	Master in Mechanical Engineering Automotive
PISO	-	Pressure Implicit with Splitting of Operator
RMS	-	Root Mean Square
UDF	-	User-Defined Functions
UTeM	-	Universiti Teknikal Malaysia Melaka

## CHAPTER 1

### INTRODUCTION

#### 1.1 Background

The fogging and defogging phenomena have attracted various research interests because of the phenomena relation with the visibility problems. The automotive windshield fogging phenomenon occurs when the condensation of the atmospheric humid air happens on cold windshield surface. D'Agaro (2006) states that the condensation of the atmospheric humid air creates water layer that is made up of small water droplets on the windshield and causes a clouded appearance due to the scattering of the light thus, reduces the visibility of the driver which leads to visibility problem which is shown in Figure 1.1. Liess et al. (2005) states that the condensation water vapour on the windshield happened when the outside temperature from the vehicle is low which this phenomenon implies the need of warm air. However, the amount of warm air enters to cabin or directed to the windshield is very small after starting the engine.



Figure 1.1: Fogging on automotive windshield (Hembree, 2016)

Based on Figure 1.1, it is clearly shows that the mist or fogging reduces the visibility which the driver cannot see clearly or does not has adequate visibility to watch and also spot everything in the sight in front of the vehicle. The fogging phenomenon is a major concern as it leads to accident risks. Therefore, the defogging of the automotive windshield should occurred quickly to helps regain adequate visibility and thus, enhance the safety of driver and passengers.

Aroussi et al. (2001) states that the studies and researches related to the defogging phenomenon on the automotive windshield started since decades ago whether the research is by experiment, simulation or both. In this project, the phenomenon of defogging on the windshield is numerically investigated using computational fluid dynamics (CFD) software called ANSYS-Fluent that can simulates fluid flow in virtual environment. The obtained data such as water layer thickness and temperature distribution of the water layer film on the windshield can be utilized to improve the defogging process such as changing the design the vehicle air flow system.

## **1.2 Problem Statement**

Fogging phenomenon that occurs on the automotive windshield is a major problem that increases the possibility of accident to happen due to lack of visibility. This intrigues the needs to understand and research on the defogging phenomenon, realizing that the fogging phenomenon as a potential of harm during driving. Defogging is the solution for the fogging phenomenon but it is unpredictable with the constant changes of the surrounding variables such as humidity and temperature of the surrounding air. Therefore, there is a necessity to investigate the defogging phenomenon on a fogged windshield by experiment or simulation. This helps in better understanding of the manner of the defogging phenomenon and also identifies the means to enhance the phenomenon so that the fogged windshield clears up in a more efficient way and faster.

## **1.3 Objectives**

The purposes of the study are to establish a numerical model for investigation of the defogging phenomenon on the automotive windshield and to obtain the critical data in the defogging phenomenon such as velocity distribution in the model and water layer thickness and temperature distribution of the water layer film on the windshield.

## 1.4 Scope

This study focuses on establishing a numerical model using ANSYS Fluent version 17.1 as software tool for simulation where the defogging phenomenon on the automotive windshield is investigated. The numerical model is in three dimensional (3D). Critical data of the phenomenon such as velocity distribution in the model, water layer thickness and temperature distribution of the water layer film on the windshield in the simulation are determined in six cases where the manipulative variables are the relative humidity of the fluid flow through the inlets which carry out in three cases of 54%, 73% and 91% relative humidity and the size of the inlet which also carry out in three cases of  $0.0025 \text{ m}^2$ ,  $0.01 \text{ m}^2$  and  $0.0225 \text{ m}^2$  area of the inlets.

## CHAPTER 2

### LITERATURE REVIEW

#### 2.1 Fogging

Fog can be defined as the cloudlike mass or minute water layer droplets on the surface of the automotive windshield. Leriche et al. (2015) states that the trigger of the condensation of the water vapour or the fogging phenomena starts when a cold surface exposed to warmer humid air where if the surface which is the windshield is colder than dew point temperature. Fog on the windshield causes rays of light to scatter because of the difference of water and air refractive index where it creates a blurred view across the windshield thus reduce the visibility of the driver especially which is dangerous while driving.

Fogging also can occur when vehicle enters a tunnel. Bopp and Peter (2006) found that the temperature on the surface of the windshield can drop below dew point and leads to the condensation process on the surface because of the cooling of the tunnel air on the vehicle windshield surface. Zolet et al. (2011) states that fogging or misting phenomena can happened inside or outside of the vehicle windshield but the fog on the outside area of the windshield can be easily wiped by turning on the wiper whereas the fog on the inside area of the windshield is more complicated and needs longer time to demist which requires a better and improve heating, ventilation and air conditioning (HVAC) system to shorten the defogging time.

#### 2.2 Condensation

Condensation is a phenomenon or occurrence that happens when the temperature of the vapour decreased lower than the saturation temperature of the vapour. The vapour that comes in contact with solid surface which the solid surface temperature is lower than the saturation temperature of the vapour allows the condensation phenomenon to happen. The same occurrence will also happen when the vapour is exposed to the free surface of a liquid or in a gas which the temperature of the liquid or gas is lower than the saturation temperature of the vapour where fog will formed when the droplets of liquid are suspended in the gas.

Fayazbakhsh and Bahrami (2013) state that fogging or mist formation is the condensation process that starts with the forming mist over the surface. Then, when the surface continuously exposed to humid air, condensation process continues in form of dropwise condensation where the droplets become bigger and coalesce of each droplets lead to formation of big drops of water as shown in Figure 2.1. This causes the increase weight of the drops and they overcome the surface tension forces which lead the droplets to become unstable and flow one after another. When the temperature differences between the surface and the moist air become higher together with humidity of the air, the condensation rate will keep increasing which then lead to film condensation. This shows that there are two types of condensation which are the dropwise condensation and film condensation.

Dropwise condensation has greatly higher heat transfer rates than film condensation because when big drops of water is formed, the droplets slip down under with the influence of gravity that causing the wall or the surface become clearer and more surface area exposed to the vapour that allows high heat transfer rate because the heat transfer resistance medium decreases. The situation is vice versa for film condensation where the present of the liquid film between the solid surface and the vapour become the heat transfer resistance. The further condensation of the vapour along the direction of the flow leads to increasing thickness of the liquid film which results in inclining of the heat transfer resistance. However, condensation in various heat transfer applications are assumed to be film condensation as it is conventional and also because the dropwise condensation usually does not remains extensively and will change to film condensation later on.

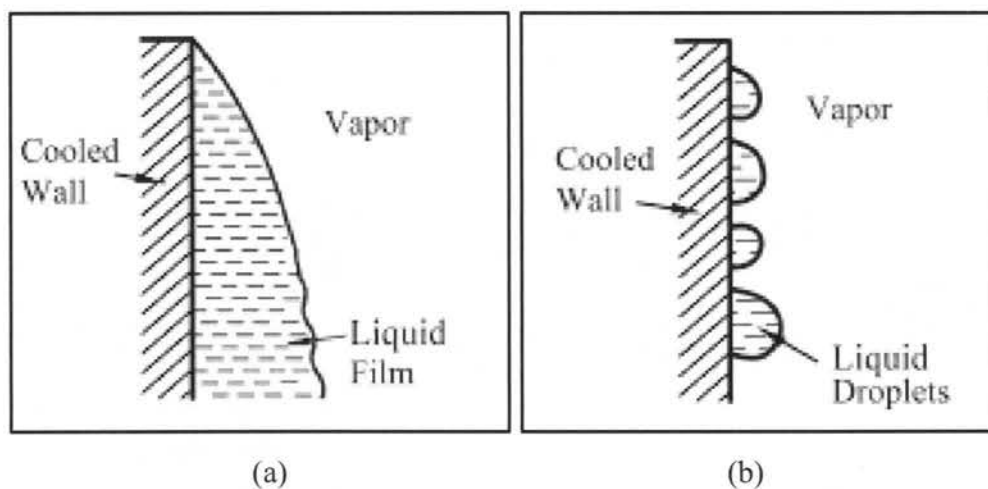


Figure 2.1: Condensation types (a) Film condensation (b) Dropwise condensation  
(Faghri and Zhang, n.d.)

2.3 Defogging

Shojaeefard et al. (2015) states that defogging or demisting of the automotive windshield is one of the important jobs of HVAC system which shows the needs of improving HVAC system performance besides to improve and save more vehicle energy that is crucial for present vehicles especially such as battery electric vehicle (BEV) and hybrid electric vehicle (HEV). The research on windshield defogging started by Stouffer and Sharkitt (1987) that invented a device called fluidic oscillator which is an air oscillator that is used to improve the flow distribution on the windshield by providing the air to windscreen with satisfactory back pressure to control ports.

Next, on year 1990, thermography was used by Dugand and Vitali (1990) and Carignano and Pippione (1990) where the two studies introduced thermography to analyse windshield defogging and also to identify thermal fields on emitting surface which allows changes in outlets and pipes as well as examine for solutions at ambient temperature. The thermography setup can be seen in Figure 2.2. They also implied a method to process the images acquired and suggested many methods to enhance windshield defogging process. Next, Lee et al. (1994) used Integrated Computer-aided Engineering and Manufacturing Computational Fluid Dynamics (ICEM-CFD) software which is CFD software for their research of simulation of the windshield de-icing system. Computer-aided drawing (CAD) software was utilized to draw the full configuration of a vehicle and it is exported to the ICEM-CFD where then, the mesh was generated and multi-domain approach is used to assemble the model.

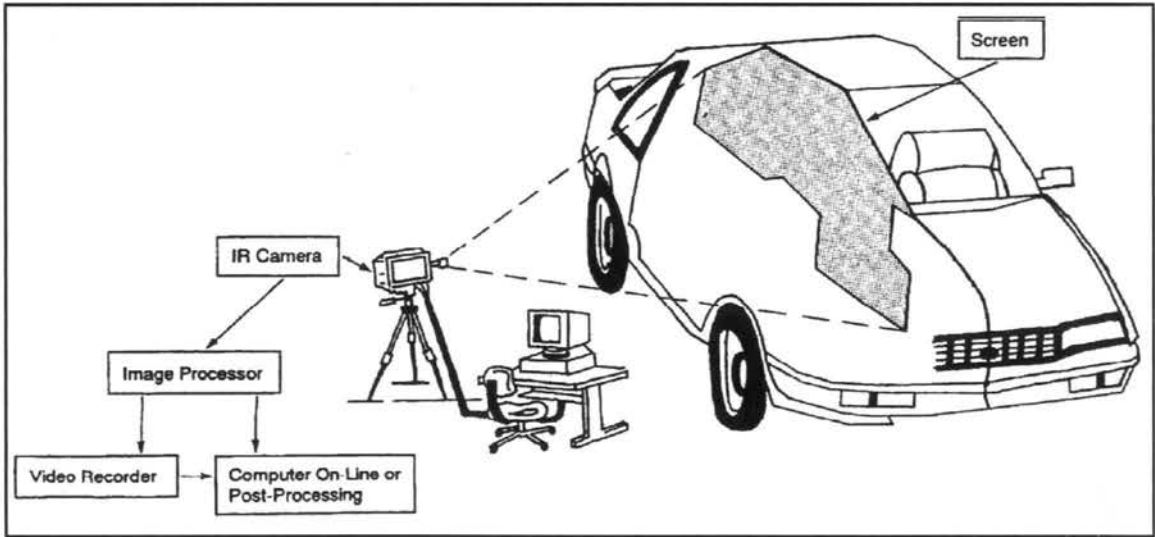


Figure 2.2: Thermography setup (Burch et al., 1993)

Later, Currie (1997) studied on the calculation of the internal field flow by using numerical simulation. The study showed the capability of CFD to estimate the internal aerodynamics of vehicle where the computational model used has comprised all air ducting components to observe the flow field in passenger compartment, the airflow of the HVAC system and also enabled the analysis of each parts of the HVAC system. In the same year, a study by Brewster et al. (1997) showed the use of CFD software which is STAR-CD by Computational Dynamics-Analysis & Design Application Company Ltd (CD-adapco) to build a 3D windshield model and analysing the ice building phenomena on windshield. Three years then, Aroussi et al. (2000) studied on full-scale vehicle heat transfer and prevailing fluid flow on windshield and represented few methods which elevating effective defogging and de-icing phenomena through passive manners and utilizing the existing vehicle HVAC system.

Defogging and de-icing are different but they have the same concept of removing or eliminating the medium on the windshield that reduce or obstruct the vision of the driver which is crucial as adequate visibility of the driver is important to avoid unnecessary risks. Roy et al. (2005) states that two-phase fluid thermal system which to remove a thin ice layer on the outer side vehicle windshield surface is the de-icing process.

Jet impingement is one of the practical applications in the defogging system of automotive windshield. Kumar and Roy (2005) state that in the recent years, there are substantial interest on the research of jet impingement heat transfer implementation in defogging system. Roy et al. (2002) states that the jet impingement has many elements that can be investigated to optimize the impinging air jet flow performance to clear the fog or defogging process where the vehicle defroster nozzle should accomplished in produce an airflow which can distributes to all over the inside surface part of the windshield. However, Roy et al. (2002) found that the real flow that exiting from the defroster nozzle has its own complication which is difficult and challenging to create the numerical solution. Besides that, Unverdi (2010) states that bus also has typical defogging system which is impinging the heated air jets departing the vents generating mass transfer or phase change, installed on a duct and is placed at the front side windows and on the inside surface part of the windshield that is under the dashboard. In addition, besides defogging system, Elnajjar et al. (2013) states that jet impingement is used on various other engineering applications such as optimizing and enhancing heat transfer, drying, glass tempering and cooling.

2.4 Convection

Convection is one of the three basic heat transfer mechanisms where the other two are conduction and radiation. Convection is a process where heat transfer occurred between two mediums which are between solid and fluid where the fluid is in motion or flowing which is different from conduction where heat transfer happened between mediums but in absence of motion of fluid. However, pure conduction heat transfer also happened during the fluid motion on a solid surface at fluid layer next to the surface where the motion of fluid approach to a complete stop on the surface due to no-slip condition as the fluid adhere to the solid surface when contact directly because of the viscous effect which is shown in Figure 2.3.

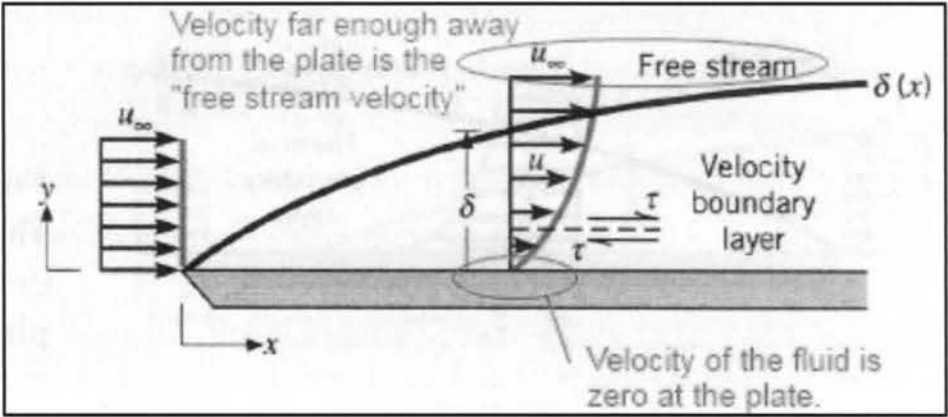


Figure 2.3: Boundary layer of velocity progression on flat surface (Scienceofdoom, 2011)

After that, the conducted heat is carried and gradually shifted to fluid flows at the outside boundary layer,  $\delta$ . Boundary layer is known as the flow area that is adjacent to the surface or wall where the velocity gradient and viscous effect are notable. Boundary layer is zero when the fluid begins flowing on the solid surface and continues to rise when the friction between the fluid layer and the solid surface also increase due to more fluid is being slowed as the distance of the flow increases in the flow direction. The rate of heat transfer can increase through convection when the fluid flow velocity increases. The rate of convection heat transfer coefficient is proportional to the difference of temperature which can be expressed as Newton’s law of cooling. Jiji (2009) state that the Newton’s law of cooling can be calculated theoretically using Equation (2.1) if the parameters in the equations are known. The heat transfer coefficient of convection is different along the flow course where the local convection heat transfer coefficients across the total surface are averaged to get the mean convection heat transfer coefficient of the surface.