COMPARISON OF LAMINAR AND TURBULENT MODEL OF WATER FLOW IN MICROCHANNEL

MOHAMED FIQRI BIN AHMED FAUZI

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering (Hons)

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2018

DECLARATION

I declare that this thesis entitled "Comparison of Laminar and Turbulent Model of Water Flow in Microchannel" is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :..................................

Name of Supervisor :..................................

Date :..................................

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Hons).

Signature	: ....
Name of Supervisor	: Dr. Ernie Binti Mat Tokit
Date	: ………………………

DEDICATION

This thesis is dedicated for the sake of Allah, my Creator and my Master, my great teacher and messenger, Muhammad (May Allah bless and grant him), who taught us the purpose of life. Other than that, this thesis is dedicated to Dr. Ernie Binti Mat Tokit how is always give me guidance for me to finish my thesis, she always makes time for me to educate me. Not to forget this dedication goes to my family that never tired of helping me and give me moral support to finish this study, colleagues and myself. Thank you.

ABSTRACT

A flow can be laminar, turbulent or transitional in nature. This become a very important classification of flow. Laminar and turbulent flow can be test to get the different of the result. The objective of this study are to find temperature distribution, mesh independent test, entrance length and find Nusselt number and the different between laminar and turbulent flow. Temperature distribution can be conduct with running the simulation that the domain have been create using software. Software that have been using in this experiment to draw the domain is GAMBIT software and that was one of the method in this study. The simulation need to consider the material use and boundary condition. Then run the simulation with iterate is set to 1000 and when the worlds converge appear the simulation will stop and the running of simulation is done. To make the simulation is valid, the numerical code is verified by contrasting or comparing it with available analytical solution or widely accepted numerical results. The largest deviation between simulation result from Qu and Mudawar result is 13.1%. The simulation is considered as valid since the simulation is showing the identical pattern. Temperature distribution can be seen in the contour where the temperature at the inlet, middle and outlet is different. The finding shows that temperature for the laminar and turbulent flow is different. Analyses data showed that turbulent flow temperature is higher than laminar. The range of hydraulic diameter are between 8.658×10^{-5} to 2.52×10^{-5}. Average range for Nusselt number in this project is between 4 to 40 Nusselt numbers. Turbulent method was using k-epsilon in simulation.

ABSTRAK

Aliran boleh bersifat laminar, bergolak atau transisi. Ini menjadi klasifikasi aliran yang sangat penting. Laminar dan aliran bergelora boleh diuji untuk mendapatkan hasil yang berbeza. Objektif kajian ini adalah untuk mencari pengagihan suhu, uji bebas ujian, panjang pintu masuk dan mencari nombor Nusselt dan perbezaan antara aliran laminar dan turbulen. Pengedaran suhu boleh dijalankan dengan menjalankan simulasi yang domain telah dibuat menggunakan perisian. Perisian yang telah menggunakan dalam eksperimen ini untuk menarik domain adalah perisian GAMBIT dan itu adalah salah satu kaedah dalam kajian ini. Simulasi perlu mempertimbangkan penggunaan bahan dan syarat sempadan. Kemudian jalankan simulasi dengan iterate ditetapkan ke 1000 dan apabila dunia menumpu muncul simulasi akan berhenti dan menjalankan simulasi dilakukan. Untuk membuat simulasi itu sah, kod berangka diverifikasi dengan membezakan atau membandingkannya dengan penyelesaian analisis yang tersedia atau keputusan berangka yang diterima secara meluas. Penyimpangan terbesar antara keputusan simulasi dari Qu dan Mudawar ialah 13.1%. Simulasi dianggap sebagai sah kerana simulasi menunjukkan corak yang sama. Pengagihan suhu boleh dilihat dalam kontur di mana suhu di salur, tengah dan salur adalah berbeza. Hasilnya menunjukkan bahawa suhu bagi laminar dan aliran bergelora adalah berbeza. Analisis data menunjukkan suhu aliran turbulen lebih tinggi daripada laminar. Julat diameter hidraulik adalah antara 8.658 x 10-5 hingga 2.52 x 10-5. Julat purata bagi nombor Nusselt dalam projek ini adalah antara 4 hingga 40 nombor Nusselt. Kaedah gelora menggunakan k-epsilon dalam simulasi.

ACKNOWLEDGEMENT

First and foremost, I would like to express my praise to Allah S.W.T for giving me the opportunity and strength to finish my final year project successfully. I would like to convey my gratitude to my supervisor, Dr Ernie Binti Mat Tokit for his guidance, encouragement and support along the way the project has taken place. He helped me a lot and giving me the space to grow myself independently.

I also would like to thank to my parents who always helped with moral support and in financial support for me to finish my task. .Not forgotten, I would like to thank to Mechanical Engineering Faculty of Universiti Teknikal Malaysia Melaka (UTeM) for providing a good facility to complete my project.

Last but not least, a huge thank to my colleagues who helped me through up and down and also my family for their words of encouragement that keeping me focus on completing this project.

CONTENT

LIST OF FIGURES

FIGURE	TITLE	PAGE

LIST OF TABLE

LIST OF ABBEREVATIONS

HVAC	Heating Ventilation Air Conditioning
GAMBIT	Geometry And Mesh Building Intelligent Tool
CFD	Computational Fluid Dynamic
CAD	Computer Aided Design
CATIA	Computer Aided Three-dimension Interactive Application
ANSYS	Analysis System Simulation Software
CPU	Central Processing Unit
GUI	Graphical User Interface
Re	Reynold Number
GEO Mesh	Geometric mesh
SAT files	Standard ACIS Text
ACIS	Andy, Charles, Ian's System
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing

LIST OF SYMBOL

c	=	Damping coefficient
ρ	=	Density
μ	=	Viscosity
ν	=	Kinematic Viscosity
R_{air}	=	Specific gas constant
k	=	Specific heat ratio
cp	=	Specific heat
cv	=	Specific heat
c	=	Speed of sound
υ	=	maximum velocity
L	=	linear dimension
t	=	denotes the time.
μ	=	dynamic viscosity of the fluid
Re	=	Reynold number
d_h		hydraulic diameter of channel

CHAPTER 1

INTRODUCTION

1.1 Background

Boundary layer can define the thickness δδ to be the distance across a boundary layer from the wall to a point where the flow velocity has essentially reached the free stream velocity u_0. Prandtl idea was to divide the flow into two regions. One of that are an outer flow region that is inviscid and or irrotational and an inner flow region called a boundary layer a very thin region of flow near a solid wall where viscous forces and rotationally cannot be ignored. The concept of the boundary layer are implies that flows at high Reynolds numbers that can be divided up into two unequally large regions. In the bulk of the flow region, the viscosity can be neglected and the flow corresponds to the inviscid limiting solution. This is called the inviscid outer flow. The second region is the very thin boundary layer at the wall where the viscosity must be taken into account. The boundary layer approximation corrects some of the major deficiencies of the Euler equation by providing a way to enforce the no-slip condition at solid walls. Hence, viscous shear forces can exist along walls, bodies immersed in a free stream can experience aerodynamic drag and flow separation in regions of adverse pressure gradient can be predicted more accurately.

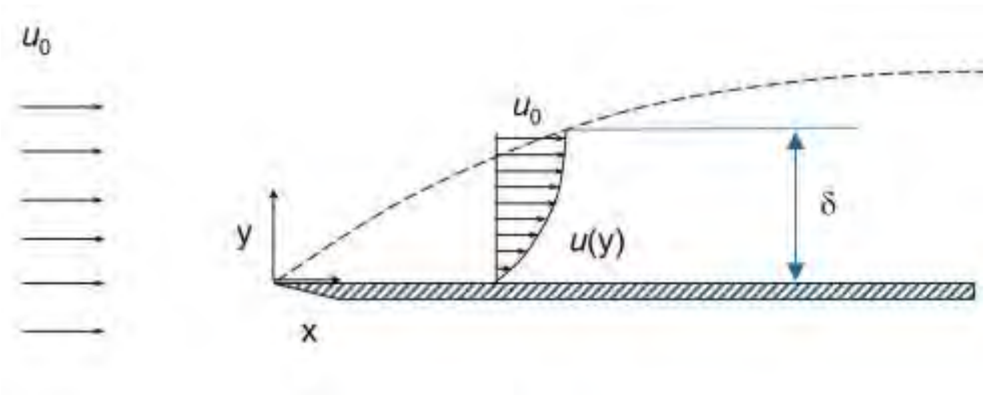

Figure 1.1: Schematic drawing depicting fluid flow over a flat plate.

Has basically achieved the free stream speed converts into a tradition. This thing can happens when the speed has the estimation of 99% of u_o. Fundamentally, arrange at the divider and look outside it until the point when the speed is that of the unperturbed stream.

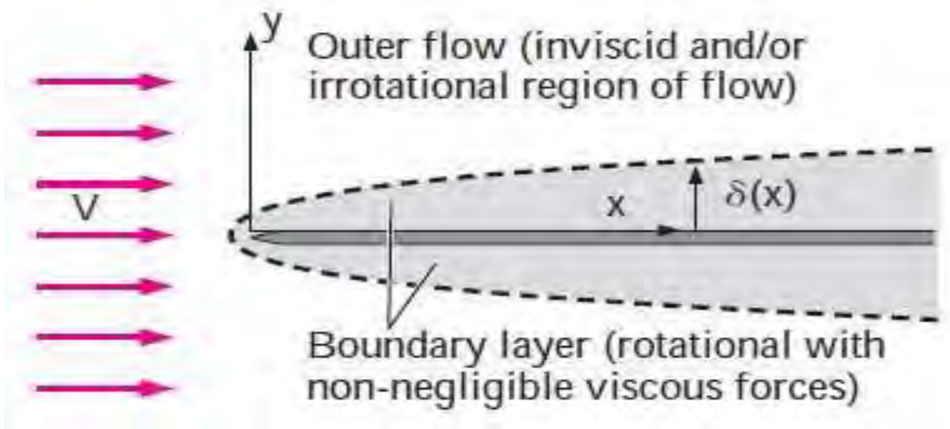

Figure 1.2: Prandtls boundary layer concept splits the flow into an outer flow region and a thin boundary layer region

The same boundary layer and observe that there is a reduction in the flow rate due to its presence. That reduction is symbolized by the shaded area in the figure. The boundary layer appears due to the shear stress that occurs in the fluid between the layers of the fluid and between the fluid and the wall. The flow were frictionless inviscid would not lose that mass flow rate. A similar boundary layer and watch that there is a diminished in the stream rate because of its essence. That diminished is symbolized by the shaded region in the figure. The limit layer shows up because of the shear push that happens in the fluid between the layers of the fluid and between the fluid and the divider. If the flow were frictionless inviscid it would not lose that mass flow rate. To obtain in an inviscid flow, an equivalent situation regarding the mass flow rate, move the boundary to compensate for the mass deficit.

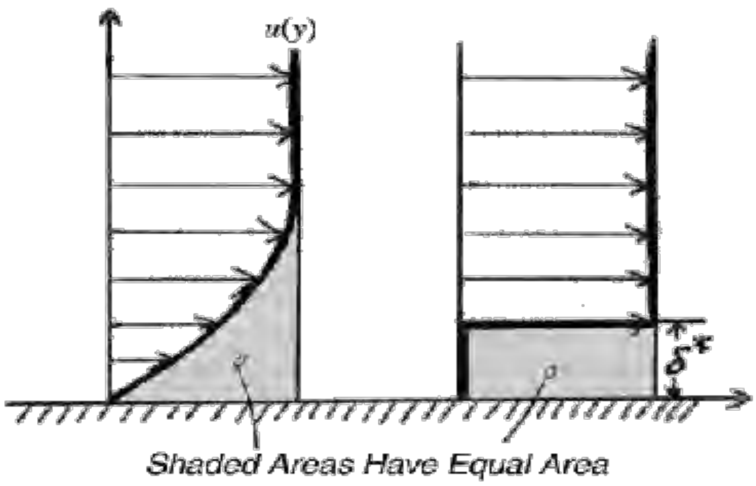

Figure 1.3: The boundary to compensate for the mass deficit

The boundary need to move with a distance such that the mass flow loss from moving the boundary equals the loss in the boundary layer. At the end of the day, the two shaded regions must be equivalent. The distance by which we move the boundary in an inviscid flow to adjust for this no friction assumption is the uprooting thickness. There is something more, it do not lose just mass stream rate when not thinking about the limit layer, but rather it additionally lose force. The separation to represent the loss of energy in the limit layer, when contrasted with that of potential flow. Move the limit by a specific separation can dispose of the limit layer, flow without thickness, and still have a similar force. This separation is the energy thickness.

In this study, the three-dimensional fluid flow and heat transfer in a rectangular microchannel heat sink are analyzed numerically using water as the cooling fluid. The heat sink consists of a 1cm2 silicon wafer. The micro-channels have a width of 57 μm and a depth of 180 μm, and are separated by a 43 μm wall. For the micro-channel heat sink investigated, it is found that the temperature rise along the flow direction in the solid and fluid regions can be approximated as linear. For a relatively high Reynolds number of 1400, fully developed flow may not be achieved inside the heat sink. Increasing the thermal conductivity of the solid substrate reduces the temperature at the heated base surface of the heat sink, especially near the channel outlet. (Qu & Mudawar, 2002).

1.2 Problem statement

Critical Reynolds number is important, the critical Reynolds number at which the flow downstream of an orifice or nozzle in a pipe becomes turbulent is an important parameter as it is the demarcating point between purely laminar flow and re-laminar flow.

The value of the critical Reynolds number is estimated for sharp-edged orifices, quadrant-edged orifices and long radius nozzles from indirect evidences using mean flow measurements. The Reynolds number is defined as the ratio of inertial forces to viscous forces and consequently quantifies the relative importance of these two types of forces for given flow conditions. Reynolds numbers frequently arise when performing scaling of fluid dynamics problems and as such can be used to determine dynamic similitude between two different cases of fluid flow. Laminar flow occurs at low Reynolds numbers, where viscous forces are dominant and is characterized by smooth, constant fluid motion. Turbulent flow

Occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortices and other flow instabilities. Viscosity is a physical mechanism for smoothing flow variations, there can be a problem differentiating between numerical and physical smoothing. This is especially important when critical Reynolds number situations are encountered, because they require an especially accurate estimate of viscous stresses.

For a laminar boundary layer growing on a flat plate, boundary layer thickness δ is at most a function of V, x, and fluid properties ρ and μ. It is a simple exercise in dimensional analysis to show that δ/x is a function of R_{ex}. In fact, it turns out that d is proportional to the square root of R_{ex}. Infinitesimal disturbances in the flow begin to grow and the boundary layer cannot remain laminar and begins a transition process toward turbulent flow. For a smooth flat plate with a uniform free stream, the transition process begins at a critical Reynolds number, $Re_{x\ critical} \cong 1 \times 105$, and continues until the boundary layer is fully turbulent at the transition Reynolds number, Re_x, transition $\cong 3 \times 106$. For laminar plate boundary layers the boundary layer thickness can easily be estimated as follows in the boundary layer the inertial forces and the friction forces are in equilibrium. The boundary layer on a plate is laminar close to the leading edge and becomes turbulent further downstream, whereby the position of the transition point x_{crit} can be determined by the critical Reynolds number $Re_{x\ crit}$ given.

1.3 Objective

The objectives of this project are as follows:

1. To determine temperature distribution
2. To find mesh independent test of water flow using laminar and turbulent flow
3. To determine the entrance length
4. To determine the Nusselt number of water flow using laminar and turbulent flow.

1.4 Scope of project

The scopes of this project are:

1. For microchannel, the range critical Reynold number is between 140 to 1000
2. The fluid use is water
3. The hydraulic diameter is 8.658×10^{-5} m for rectangular microchannel
4. The turbulent used is k-epsilon
5. The Reynolds number of water flow is 1000

1.5 General Methodology

The actions that need to be carried out to achieve the objectives in this project are listed below.

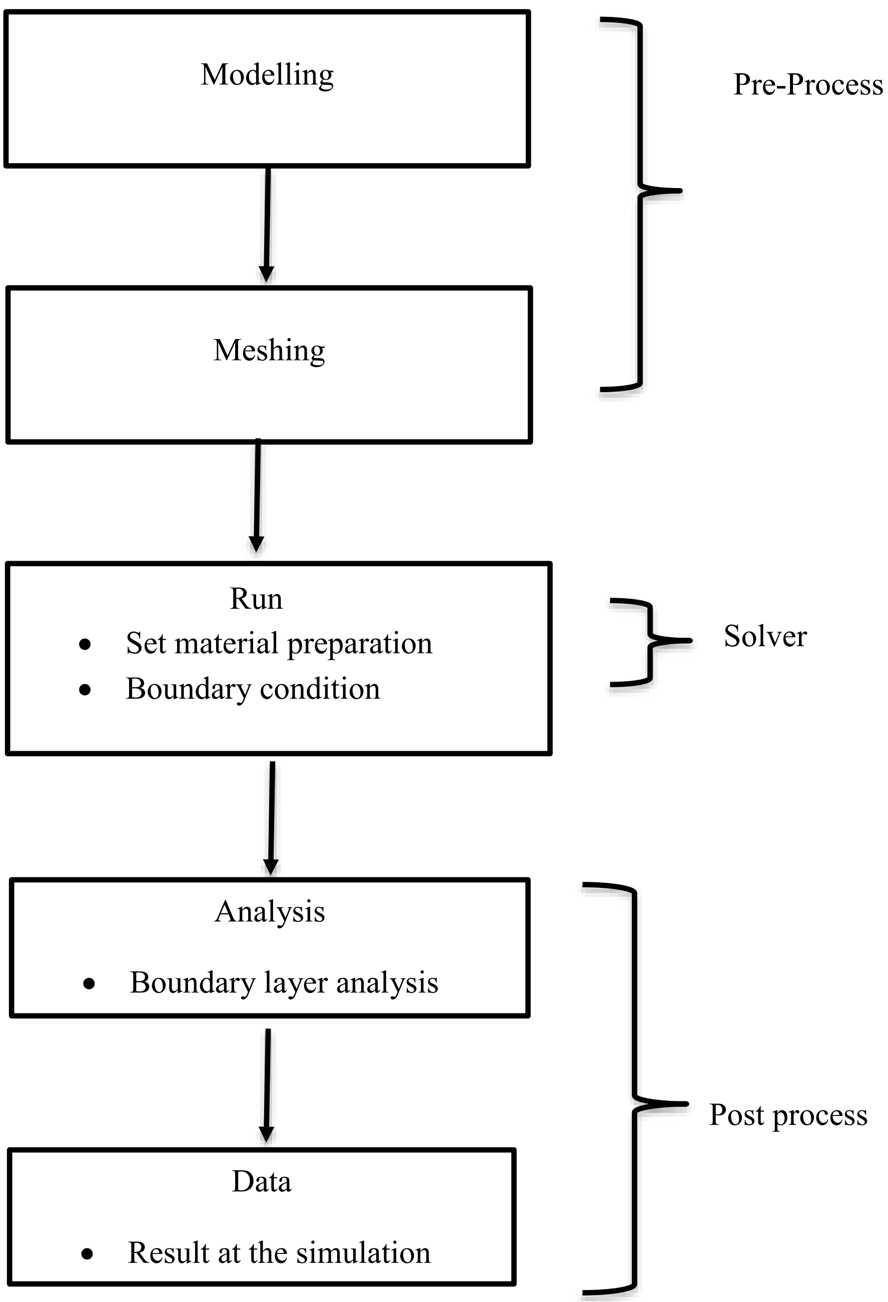

Figure 1.4: Flow chart of the methodology

1. Literature review

   Books, journals, article and all other alternative reference for this project will be review.
2. Drawing (pre-processing)

   The microchannel
3. Simulation

   Simulation of the water velocity profiles using fluent software based on the several of Reynolds number.
4. Report writing

   A report on this study will be written at the end of the project.

CHAPTER 2

LITERATURE REVIEW

2.1 Reynolds Number

Critical Reynolds Number in Pipe Flow by Stephen Mirdo (Stephen Mirdo 2010). The Reynolds number (Re) is an important dimensionless quantity in fluid mechanics used to help predict flow patterns in different fluid flow situations. At low Reynolds numbers flow has a tendency to be commanded by laminar sheet-like flow, however at high Reynolds numbers turbulence comes about because of contrasts in the fluids speed and course, which may once in a while converge or even move counter to the general heading of the stream vortex ebbs and flows.

Hiroaki Nishikawa, Yi Liu (Hiroaki Nishikawa & Yi Liu 2017) hyperbolic advection diffusion schemes for high Reynolds number boundary layer problems. Reynolds number has wide applications, running from fluid flow in a pipe to the entry of air over an air ship wing. It is utilized to predict the progress from laminar to turbulent flow, and utilized as a part of the scaling of comparable however unique measured flow circumstances. The expectations of the beginning of turbulence and the capacity to ascertain scaling impacts can be utilized to help anticipate liquid conduct on a bigger scale, for example, in neighborhood or worldwide air or water development and in this manner the related meteorological and climatological impacts.

Yong-Hui Wang et al. (2017) stated that the Reynolds number is the ratio of inertial forces to viscous forces within a fluid which is subjected to relative internal movement due to different fluid velocities, in which is known as a boundary layer in the case of a bounding surface such as the interior of a pipe. A similar effect is created by the introduction of a stream of higher velocity fluid, such as the hot gases from a flame in air. This relative movement results in fluid friction, which is a factor in building turbulent flow. Balancing this effect is the viscosity of the liquid, which it increases, gradually inhibit turbulence, as more kinetic energy is absorbed by the viscous liquid. The Reynolds number calculates the relative importance of the two types of forces for a particular flow condition and is a guide when turbulent flows will occur under certain conditions.

S. A. Si Salah, E. G. Filali, S. Djellouli (2006). The ability to predict the start of turbulent flow is an important design tool for equipment such as a pipe system or aircraft wing, but the Reynolds number is also used in fluid dynamics scale scales, and is used to determine the dynamic equations between two types of fluid flow cases, such as between aircraft models, and its full size version. Such grading is not linear and the use of the Reynolds number for both conditions allows the scale factor to be developed. Laminar flow occurs at low Reynolds numbers, where the viscosity is dominant, and is characterized by continuous constant fluid motion. Turbulent flows occur at high Reynolds numbers and are dominated by inertial forces, which tend to produce stirring eddies, vortices and other flow instability. The Reynolds number is defined as

$$Re = \frac{\rho \upsilon \mathcal{L}}{\mu} = \frac{\upsilon \mathcal{L}}{\nu}$$

ρ is the density of the fluid . SI units is kg/m^3