PREDICTING TURBULENT FLOW IN A STAGGERED TUBE

MOHD ADIB MUIZZUDDIN BIN MOHLIS

Laporan ini dikemukakan sebagai
Memenuhi sebahagian daripada syarat penganugerahan
Ijazah Sarjana Muda Kejuruteraan Mekanikal (Termal-Bendalir)

Fakulti Kejuruteraan Mekanikal
Universiti Teknikal Malaysia Melaka

September 2007

‘Saya/Kami* akui bahawa telah membaca
karya ini dan pada pandangan saya/kami* karya ini
adalah memadai dari segi skop dan kualiti untuk tujuan penganugerahan
Ijazah Sarjana Muda Kejuruteraan Mekanikal (Termal-Bendalir)’

Tandatangan :…………………………………..
Nama Penyelia I:………………………………..
Tarikh :………………………………………….

Tandatangan:……………………………………
Nama Penyelia II:………………………………
Tarikh:………………………………………….

Saya akui laporan ini adalah hasil kerja saya sendiri kecuali ringkasan dan petikan yang tiap-tiap satunya saya telah jelaskan sumbernya”

Tandatangan :……………………………..
Nama Penulis :……………………………
Tarikh :……………………………………

ACKNOWLEDGEMENT

Alhamdulillah, thanks to ALLAH because I have completed my Projek Sarjana Muda 1 without facing any major problem.

First and foremost I would like to deliver my deep appreciation and thankful my supervisor which is my Heat Transfer Subject lecturer, Encik Shamsul Bahari Bin Azraai for being such a great mentor to me. Much thank and gratitude to Encik Ahmad Kamal Bin Mat Yamin, Mechanical Engineering supervisor for Projek Sarjana Muda, for his good concern. I also would like to thanks my family for encourage and motivate me during this period of industrial training. Besides that I also would like to thank them for financial support.

Lastly, thanks to Faculty of Mechanical Engineering which has gave an opportunity to discover Mechanical Engineering Field. I hope this report will be used by other students as a reference for their project.

ABSTRACT

This report is about predicting turbulent flow in a staggered tube using Computational Fluid Dynamic software developed by Ansys which is the CFX. Either CFX, there are many non computational dynamic methods that can predict the heat transfer and flow characteristic such experimental and analytical method. But using Computer Fluid Dynamic software can save time and cost because only virtual model were create and simulate. The aims of this study are to predict the fluid flow and heat transfer characteristic in a staggered tube air and water as the fluid. The flow and heat transfer characteristic of a staggered tube a nearly the same of a single cylinder/tube. The first row characteristic is usually the same as a single tube. But for the second row and so on, the characteristic are depends on the rows that’s come first. For a single tube actually there are two boundary layers which occur at front of the tube and the back of the tube. Laminar boundary layer always developed at the front while turbulent at the back. Several geometrical parameters have been defined according to the previous research (journals) done in order to design the staggered tube geometry. The other parameters such as the fluid velocity and temperature are also defined according to the previous research. With these parameters a simulation can be made using CFX. There are several process in CFX which are the CFX Pre-processing, CFX solver and CFX post processing. The results obtain are in terms of velocity, temperature and pressure profile. From the investigation lower value of pitch used will results better heat transfer rate. When the pitch value is decrease, the pressure drop will rise and thus will increase the heat transfer rate. In engineering application higher pressure drop value will result higher power required to move the fluid through the tube bank.

ABSTRAK

Laporan ini menceritakan tentang meramal dan mentafsir aliran bergelora di dalam tiub yang disusun secara berperingkat dengan menggunakan perisian pengkomputeran aliran dinamik iaitu Ansys CFX. Selain CFX, terdapat kaedah-kaedah lain yang boleh meramal dan mentafsir pemindahan haba dan sifat aliran seperti kaedah pengujian dan analitikal. Tetapi dengan menggunakan kaedah pengkomputeran aliran dinamik, masa dan kos dapat dijimatkan dan dikurangkan berbanding dengan menggunakan dua kaedah yang disebut tadi. Ini adalah kerana hanya model maya saja yang dicipta dan di kaji. Tujuan laporan ini adalah untuk meramal dan mentafsir sifat aliran dan pemindahan haba didalam tiub berperingkat dengan menggunakan air dan udara sebagai medium aliran. Sebenarnya sifat aliran dan pemindahan haba didalam tiub berperingkat adalah hampir menyamai dengan tiub tunggal. Ini adalah kerana pada barisan pertama sifatnya adalah seperti aliran tunggal. Pada barisan kedua dan seterusnya sifatnya adalah bergantung kepada barisan yang datang dahulu. Untuk tiub tunggal, terdapat dua lapisan sempadan yang terbentuk di hadapan tiub dan dibelakang tiub. Lapisan laminar selalunya terbentuk dihadapan manakala lapisan bergelora terbentuk di belakang. Beberapa parameter geometri telah ditentukan berdasrkan kajian yang telah dijalankan oleh pengkaji-pengkaji untuk mereka geometri tiub. Terdapat juga parameter lain yang telah di tentukan seperti halaju bendalir dan suhu yang juga berdasrkan jurnal. Dengan parameter-parameter ini simulasi dapat dilakukan dengan menggunakan CFX. Terdapat beberapa process didalam CFX seperti CFX pra-proses, CFX penyelesai dan CFX selepas proses. Keputusan yang diperolehi adalah didalam profil halaju, suhu dan tekanan. Berdasarkan simulasi yang telah dijalankan, didapati nilai pitch yang rendah akan meningkatkan kadar pemindahan haba. Tetapi apabila nilai pitch berkurang, kejatuhan tekanan akan meningkat dan seterusnya meningkatkan kadar pemindahan haba. Didalam aplikasi kejuruteraan, nilai kejatuhan tekanan yang tinggi akan menyebabkan lebih banyak kuasa yang diperlukan untuk menggerakkan cecair melalui tiub-tiub tersebut.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

u_∞	=	Free stream velocity, m/s
V	=	Upstream velocity, m/s
Re_D	=	Reynolds Number (for cylinder)
C_D	=	Drag Coefficient
h	=	Heat Transfer Coefficient
D	=	Tube Diameter, m
K	=	Thermal Conductivity, W/mC.ºC
v	=	Kinematic Viscosity, m^2s^{-1}
Pr	=	Prandtl Number
$C,\ C_1\ m, n$	=	Constant
Nu_D	=	Nusselt Number
υ_s	=	Mean Fluid Velocity, m/s
ρ	=	Fluid Density, kgm^{-3}
μ	=	Dynamic Fluid Viscosity, Nsm^{-2}
L	=	Characteristic Length, m
S_T	=	Transverse Pitch, m
S_L	=	Longitudinal Pitch, m
S_D	=	Diagonal Pitch, m
T_∞	=	Ambient Temperature, °C
A_1	=	Length from tangent of two tubes in transverse, m
A_2	=	Length from tangent of two tubes in diagonal, m
T	=	Temperature, °C
L	=	Length, m
ΔT_{lm}	=	Log Mean Temperature Different
N	=	Total Number of tubes in Bank

τ_w	=	Wall Shear Stress, Nm^2
T_w	=	Wall Temperature, °C
Q	=	Heat Transfer Rate, W
η	=	Distance Normal to and Measured From Surface of Tube,m
u	=	s-component of Velocity in Boundary Layer, m/s
υ	=	η - component of Velocity in Boundary Layer, m/s
$u_{\max}$	=	Mean Velocity in the Minimum Free Cross Section of the Control Volume, m/s

LIST OF ABBREVATIONS

CFD	=	Computational Fluid Dynamic
REV	=	Representative Elementary Volume
RSM	=	Reynolds Stress Model
RMS	=	Root Mean Square
2D	=	Two Dimension
3D	=	Three Dimension
CAD	=	Computer Aided Design
FEM	=	Finite Element Method
DNS	=	Direct Numerical Simulation
LES	=	Large Eddy Simulation

CHAPTER I

INTRODUCTION

This chapter will explained briefly about predicting turbulent in a staggered tube using computational fluid dynamic, the objective of this study, scopes and the problem statements.

1.1 Predicting Turbulent in a Staggered Tube

The earliest numerical solution in studying steady flow across a circular cylinder was reported during 1933. Since that early work, many studies of flow and heat transfer across tubes and within tube banks have been done using CFD (Computational Fluid Dynamic) software. Heat transfer and flow characteristic prediction across cylinder are important in various engineering aspect and have been presented in many related engineering application. This have been an active research because it have many benefits in practical of heat exchanger tube bundles, flow across overhead cables and for nuclear power plant cooling system.

To obtain the flow characteristic and heat transfer around tube bundles, Navier Stokes and Energy equation can be used to calculate them. Better degree of approximation can be obtain depending on many factors including the solution method, mesh size, boundary condition and the stability and convergence criteria. Although there are many experiment and data that had been done and collected by previous researchers, it is yet possible to get a clear view about the flow and heat transfer process across tube bundles because of its complex geometry and there are many large number of parameter involved.

In this study, CFD software will also be used to predict the heat transfer and turbulent flow across staggered tubes. CFD simulation for turbulence is harder to make rather than laminar flow because the turbulence flow field are always unsteady (random, swirling, vertical structures called turbulence eddies). There are many ways to solve the CFD calculation such as Finite Element Method, Direct Numerical Simulation and Large Eddy Simulation. Finite Element Method will be used for the CFD calculation in this study because the CFD software chooses are based on the FEM.

1.2 Objective

The objectives of this study are to predict the heat transfer and flow distribution of a staggered tube.

1.3 Scopes

- Design CFD staggered tube geometry.
- Simulate air and water through an array of heated staggered tube.

1.4 Problem Statement

To simulate a flow across staggered tube, many parameters have to be decided. These parameters will affect the simulation results. There several parameters that must be considered for predicting turbulent in a staggered tube which are:

- Longitudinal and transverse pitch
- Reynolds Number
- Thermal and wall boundary condition
- Surface Roughness (in this study it is consider a smooth tube)

- Number of rows
- Fluid properties
- Tube diameter

Figure 1.1 shows the fluid flow through a staggered tube. The flow enters at exactly normal to the tube. The detailed boundary condition will be shown in methodology chapter.

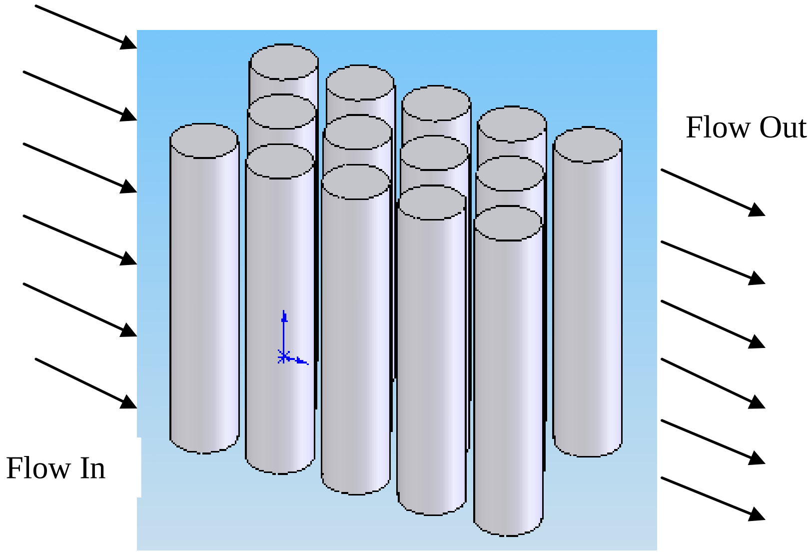


Figure 1.1: Fluid flow through a staggered a tube

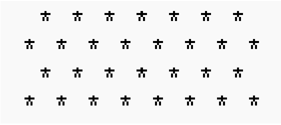
CHAPTER II

LITERATURE REVIEW

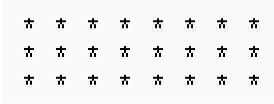
This chapter will explain in detail about the flow across cylinder and tube bank and the formula used. It also includes the previous studies that have been done by previous researchers.

2.1 Introduction

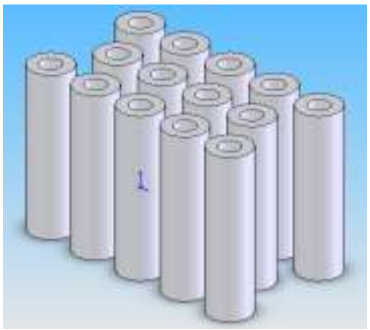
Staggered tube is a bundle/bunch of tube which are arranged in staggered. From dictionary it stated that staggered means not arranged consecutively or in a straight line [1]. This mean that the tube is not arranged in inline. The succeeding row of tubes is offset so as not to be directly behind the preceding row of tubes in the airflow direction. This type of arrangement is similar to atom arrangement which is the lattice arrangement. This will increases thermal capacity over an inline pattern at the cost of a higher air friction. Figure 2.1 (a) and 2.1 (b) shows the examples of staggered and inline arrangement using point. The isometric and top views of staggered tube are shown in Figure 2.1 (c) and (d).



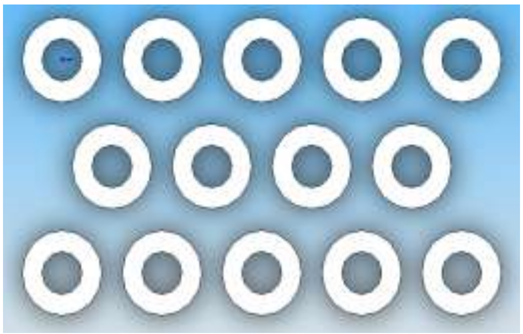
(a)



(b)



(c)



(d)

Figure 2.1: Shows the points that are arranged in staggered (a) and inline (a) while (c) and (d) shows the isometric view and top view of a staggered tube

Staggered tube is important in many industrial applications. As mentioned in the introduction chapter, it can be used in nuclear power plant, steam generation in a boiler and coil of an air conditioner. Many studies have been done because of its potential and importance in heat exchanger. In this study the staggered tube bank will be study numerically using CFD package software Ansys CFX.

2.2 Computational Fluid Dynamic

CFD is a computational technology that enables you to study the dynamics of things that flow [2]. CFD is one of the branches of fluid mechanic that uses numerical method and algorithm to solve problems that interconnect with fluid flow [3]. With CFD people can build a virtual model that represents the system and device that he/she wants to study. Then fluid flow physic and chemistry can be applied to it and computers will perform the millions of calculation that required in the simulation [2].

2.3 Flows across Cylinder

Heat transfer occurs by a cylinder cross flow is as important as heat transfer through flat plates and inside a tube. The heat transfer characteristic is affected by the development of boundary layer on the cylinder. However pressure gradient is important in the analysis and must be included because it influences the boundary layer velocity profile. It also causes a separated flow region that develops at the back of the cylinder when the free stream velocity, u_∞ is large. From figure, the fluid is bought to the forward stagnation point with increasing in pressure. From the boundary layer theory the pressure through the boundary layer is constant at any position of the body. But for cylinder case, from the forward stagnation point the pressure will decrease due to increasing of x which is the streamline coordinate and this condition will develop a boundary layer because of the favorable pressure gradient (*dp/dx<0*). The upstream velocity, *V* and u_∞ is differ from each other. The free stream velocity depends on the distance x (refer Figure 2.2) from the stagnation point. When free stream velocity equals to zero, which is at the stagnation point the adverse pressure gradient (*dp/dx>0*) will make the fluid accelerates and reaches a maximum velocity when *dp/dx=0* (at the symmetrical line of the cylinder). Note that the pressure will start increasing when it reaches the starting of the back of the cylinder and the fluid will decelerate due to the adverse pressure gradient. During this situation, the velocity is maximum and the fluid will start decreasing. As the fluid decelerates, the velocity gradient at the surface becomes zero. Figure 2.2 and Figure 2.3 show the flow across cylinder and its velocity profile that shows the velocity separation [10 and 11].

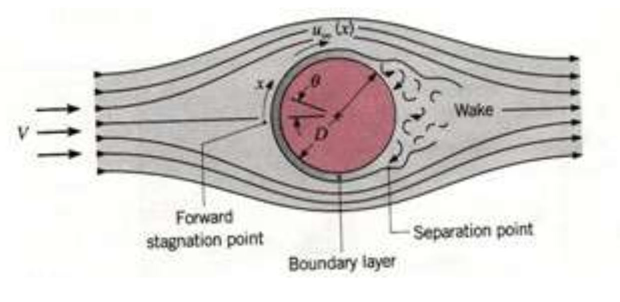


Figure 2.2: Cylinder in cross Flow

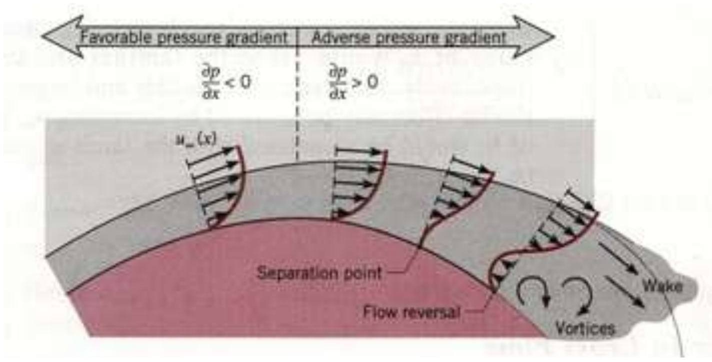


Figure 2.3: Velocity profile indicating flow separation on a cylinder in cross flow

This makes the fluid near to the surface does not have enough momentum to overcome the pressure gradient and will result a separation point ($[\delta u / \delta y]_{y=0} = 0$). Separation point is a condition where the boundary layer detaches from the surface and wake is occurring in the downstream region. Reverse phenomena of the fluid occurs as it pass the separation point [10].

Usually laminar boundary layer will develop at the front of the cylinder and turbulent boundary layer will develop at the back of the cylinder [11]. But there is also possibility that a transitional boundary layer will develop in between these two boundary layers and this is depends on the Reynolds Number. If the $\mathrm{Re}_D < 2x10^5$ the boundary layer remains laminar but when $\mathrm{Re}_D > 2x10^5$, transition boundary layer occurs and the separation will be delayed [10].

2.31 Drag Force

This repeatedly process will influence the drag force to the cylinder. It has two components which are:

- Friction Drag (due to boundary layer surface shear stress).
- Form or Pressure Drag (due to pressure differential in flow direction because of the wake formation)

A drag coefficient can be defined as:

$$C_D = \frac{F_D}{A_f(\rho V^2/2)} \tag{2.1}$$

Figure 2.4 shows the drag coefficient versus Reynolds number graph for flow across circular cylinder and sphere for comparison.

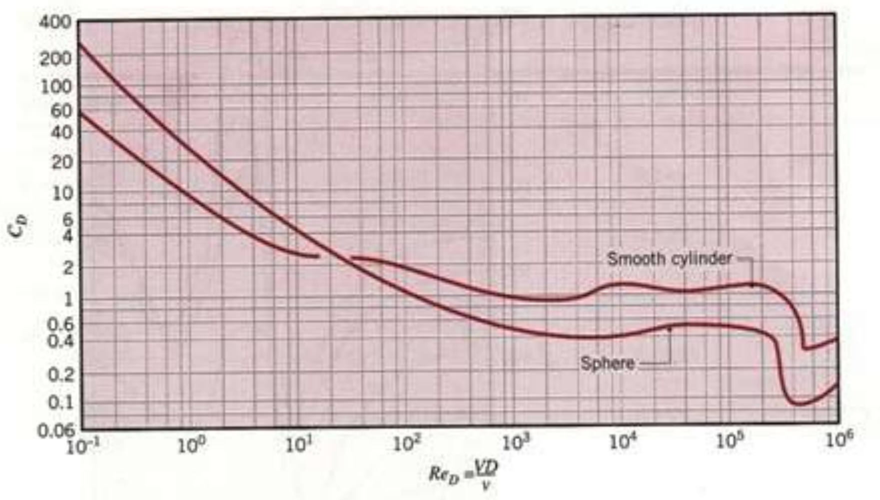


Figure 2.4: Drag corfficients for circular cylinder and sphere in cross flow

This drag coefficient is a function of Reynolds Number. For $\mathrm{Re}_D > 2$ the separation effect can be neglect because friction drag has dominated the conditions. From the graph above if the Re_D is low, the C_D will become high. This is because at low Re_d, there is no flow separation and all the drag is resulted from viscous friction. If it is high the C_D will reduce due to boundary layer transition (delay separation) and that's why will reduce the extent of wake region. The turbulent separated flow region happens when the Re_D is greater than 1000. The boundary layer becomes fully turbulent is when the Re_D is approximately equivalent to 10^5. This will result in a steeper velocity profile and extremely late flow separation [10 and 11].

2.32 Correlation for Average Heat Transfer

There are actually 3 correlation equations for average heat transfer of flow across cylinder. The equations are stated below.

1. The first equation is the easiest to use from a computational standpoint. Figure 2.5 shows the correlated data of a number of investigators for heating and cooling of air by Mc Adams [11].

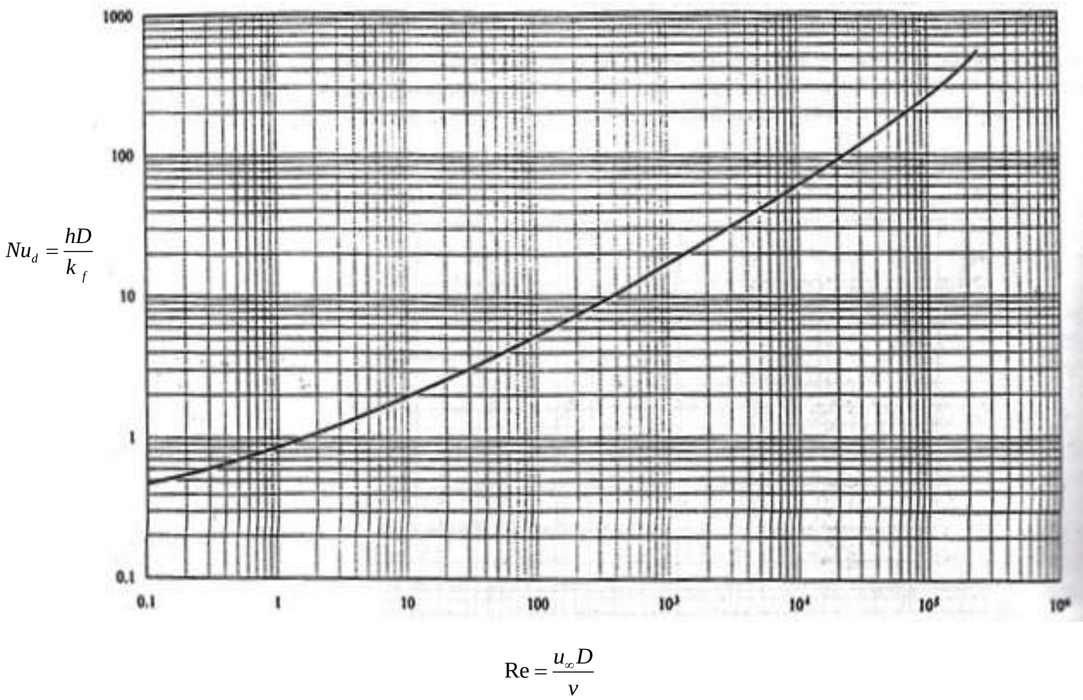


Figure 2.5: Correlation of heating and cooling for flow across cylinder [10]

The resulting correlation for average heat transfer is:

$$Nu_d = \frac{hD}{k_f} = C\left(\frac{u_\infty D}{v_f}\right)^n \mathrm{Pr}_f^{1/3} \tag{2.2}$$

The subscripts *f* means that all of the properties used are evaluated at film temperature. The Prandtl number is constant at all data because it is not included in correlation plot. The constant values of different Reynolds number used for equation (2.2) can be found in J.P Holman heat transfer book [11].