FLOW INVESTIGATION AROUND SPHERE IN OPEN CIRCUIT WIND TUNNEL

AHMAD ZHARIF RABBANI BIN ROZALI

This PSM report is submitted to Faculty of Mechanical Engineering Universiti Teknikal Malaysia Melaka in partial fulfillment for Bachelor of Mechanical Engineering (Thermal Fluid)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

27 March 2008

TABLE OF CONTENT

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

Tandatangan :...................................

Nama Penulis : **Ahmad Zharif Rabbani bin Rozali**

Tarikh :...13/05/08...................................

ACKNOWLEDGMENT

Alhamdulillah with His Mercy and Blessings, my greatest gratitude to Allah the Almighty for the chances given to me to complete this project. My first acknowledgement goes to Mr Shamsul Bahari Bin Azraai, my project supervisor. Thanks for his tremendous help, advice, inspiration and unending guidance to me until completing this research. Without him, this project will not exist. The fundamental idea behind this project is his and he never stop from giving me support and encouragement in completing this project. Without his valuable guidance, I would not have been able to achieve the objectivity of this project.

To Mr. Razmi, technican lab, thanks for your help and giving me support and idea while doing this study.

Not forgotten, to my beloved parents, Mr.Rozali bin Awang and Mrs. Aklima binti Ismail. Your love and caring really touched me. Thanks so much for giving me lifelong encouragement, inspiration and support. May Allah bless both of you.

Lastly, my acknowledgement go to my friends who really understand and willing to cooperate with me in the process of completing this project. Without your supports, I will never go this far.

ABSTRACT

The flow around a sphere at various heights above a plane boundary has been investigated. The flow was visualized by the smoke wire method in the wind tunnel. This report only covers the introduction of the research, the literature review which will state all the theory and equation needed to solve the problem statement and the method that will be use to complete this study. At the end of this study which will be continued later in Projek Sarjana Muda II, it is expected that at every different height will give different reading and pressure distribution around model. Behavior of the flow and vortices were observed during flow visualization. The surface pressure distribution on the sphere and the plane were determined by integrating the surface pressure of the sphere. The drag lift coefficients of the sphere are defined as empirical equations using dimensionless quantities. To validate this flow pattern we must use combination of numerical method using Ansys CFX and calculations of the results. The results expect that as the Reynolds Number increase the Drag number is also increase for both experimental and simulation.

ABSTRAK

Aliran pada permukaan sfera pada ketinggian yang berbeza di atas lapisan tunggal telah dikenalpasti. Aliran tersebut telah ditunjukkan dengan cara pengasapan wayar dalam terowong angin. Laporan ini hanya merangkumi pengenalan kajian, kajian literature di mana di sini segala teori dan persamaan yang di perlukan untuk menyelesaikan masalah kajian ini akan ditunjukkan dan juga cara-cara yang yang akan digunakan untuk menyiapkan kajian ini. Di penghujung kajian ini, di mana ianya akan di sambung di Projek Sarjana Muda II, ianya dijangka bahawa pada setiap ketinggian dan permukaan yang berlainan akan memberi kealainan pada corak aliran ke atas permukaan sfera tersebut. Sifat dan corak putaran di lihat sewaktu penentuan corak di lakukan. Taburan tekanana ke atas permukaan sfera ditentukan dengan cara pembahagian tekanan permukaan sfera tersebut. Daya seretan angkat bagi sfera ditentukan berdasarkan pemerhatian atau ujian dengan menggunakan kuantiti ruang yang di ukur. Bagi mengesahkan corak aliran ke atas permukaan sfera ini berkesan, perlulah kita menggunakan gabungan cara simulasi dengan menggunakan Ansys CFX dan juga pengiraan. Hasil keputusan yang dijangka adalah apabila berlaku peningkatan pada nombor Reynolds peningkatan juga berlaku pada nombor daya mengheret bagi kedua-dua eksperimen dan simulasi

LIST OF FIGURE

LIST OF SYMBOLS

u_{∞}	maximum velocity
D	sphere diameter
V	kinematic viscosity
ρ	Air density
υ_s	mean fluid velocity
L	characteristic length
μ	dynamic air viscosity
V	kinematic viscosity

LIST OF ABBREVIATION

CFD	Computational Fluid Dynamic
Ma	Mach number
TW	Tapping wire
CAD	Computer aided design
RSM	Reynolds stress model
R^3	three dimensional space
e.g.	example

CHAPTER 1

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

1.0 Introduction

Predicting heat transfer and flow behavior around sphere

There are many researchers have done a study in this topic. Each of the studies done is different in the methodology so the results obtained were different. The methodology used was experimental, and numerical/CFD. Many of them have proposed results that can be used in this topic.

A number of studies have examined the flow field around a sphere and the aerodynamic force in a uniform flow, and their fundamental properties have been demonstrated previously. For example, Achenbach [9 and 10] discussed rotating separation and the vortex shedding process for a sphere, and furthermore the effect of surface roughness and tunnel blockage were examined [11]. Taneda [8] presented the wake configuration of a sphere and Sakamoto and Haniu [6] investigated the vortex shedding from spheres in a uniform flow.

From an engineering viewpoint, the surfaces of most structures are exposed to the wind and are subject to a turbulent boundary layer and the ground effect. The spherical

structure finds application in not only gas tanks, but also in artistic structures and some types of vehicles. Therefore, in designing such structures, it is necessary to obtain experimental data concerning the flow field and the aerodynamic force on the structures. Okamoto [5] investigated the flow field of a sphere in contact with a plane and observed the wake structure and the aerodynamic force on the sphere. In the heat transfer field, a sphere was available for the heat transfer enhancement. Seban and Caldwell [7] presented the effect of a spherical protuberance on local heat transfer on a plane.

1.1 Objectives

The objectives of this study are:

- To determine the pressure distribution around sphere
- To compare the experimental result and simulation result

1.2 Scopes

The scopes of this study are:

- Set-up instrumentation on a sphere in open circuit wind tunnel
- To analyze experimental data
- Simulate air flow through a sphere using Ansys CFX software

1.3 Problems statement

Introduction

Wind tunnel testing for any other object like sphere, flat plate and other is typically performed in a boundary layer wind tunnel.
In this project, the air flow around sphere is investigated. The discussion and investigation need to be considered when doing the experiment. The problem is that

flow quality in this tunnel itself Figure 1.1 shows the air flows around a sphere at different velocity of air

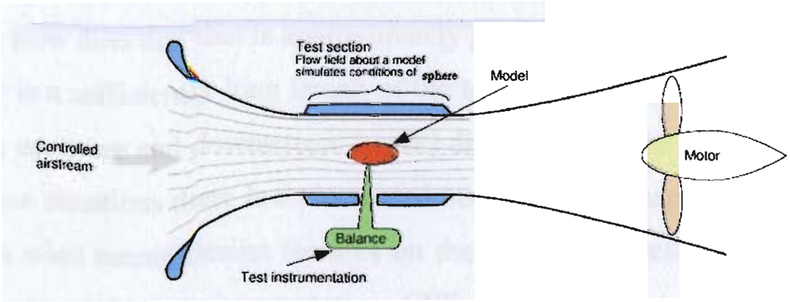

Figure 1.1 : Air flow around sphere in wind tunnel

Besides, the problem is to hold the spherical object properly. This is important in obtaining the result simulation and data obtain from the experiment conduct. These prove by reasonable answer, which is we can take two method to hold the spherical object either vertical or horizontal by using the sting as the object holder. Normally, either we choose vertical or horizontal, we also need to consider the sting interference.

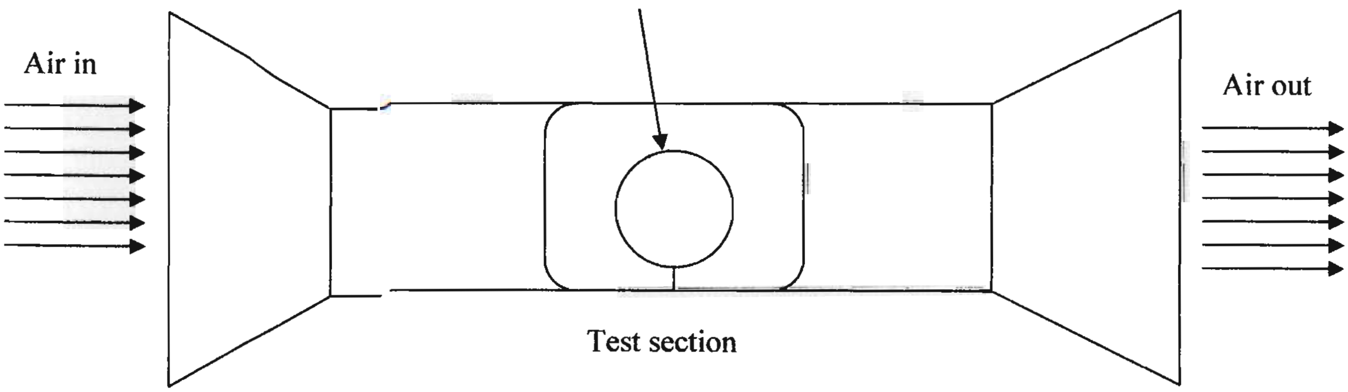

Figure 1.2 : Schematic drawing of wind tunnel

We also need to sure that these test section (wind tunnel) is low angularity, that mean flow direction that is approximately parallel to the test section center line.
Other is a sufficiently long length of the test section to allow for an accurate prediction of the upstream and downstream (wake) disturbance of the flow by the test object
In these situations there is an increased need for a thorough understanding of the impact of the wind tunnel design features on the flow in the test section and on the obtained results to guide their interpretation. CFD could then be used to obtain the information required. Generally, the numerical simulation of wind tunnel experiments is performed by modeling only the flow in the test section (rectangular computational domain) and by applying similar boundary conditions at the inlet as measured in the (empty) wind tunnel test section. However, if the above conditions are not satisfied, this ''conventional'' approach may no longer be adequate and a more extensive CFD modeling methodology is required.

CHAPTER 2

2.0 LITERATURE REVIEW

2.1 Introduction

The literature review is based on the physics of the project undertaken here, concentrating on the introduction on the sphere itself, wind tunnel and computational fluid dynamics.

2.2 Sphere body

A sphere is a symmetrical geometrical object. In non-mathematical usage, the term is used to refer either to a round ball or to its two-dimensional surface. In mathematics, a sphere is the set of all points in three-dimensional space (R^3) which are at distance *r* from a fixed point of that space, where *r* is a positive real number called the radius of the sphere.

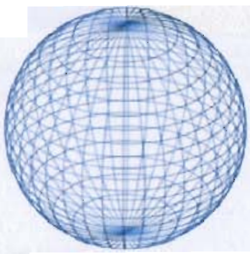

Figure 2.1: Sphere model

2.3 Wind Tunnel

Wind tunnel testing is a research tool develops to assist studying the effect of air moving over or around solid object. There are three ways that wind- speed and flow are measured in wind tunnel which is threads can attached to the surface of study objects to detect flow direction and relative speed of air flow, dye or smoke can be injected upstream into the air stream and the streamlines that dye particles follow photographed as the experiment proceeds and pitot tube probes can be inserted in the air flow to measure static and dynamic pressure.

Type of Wind Tunnel

1. Open –loop

   In an open loop wind tunnel, the intake and exhaust ends of the tunnel are not connected, but this isn't very economical

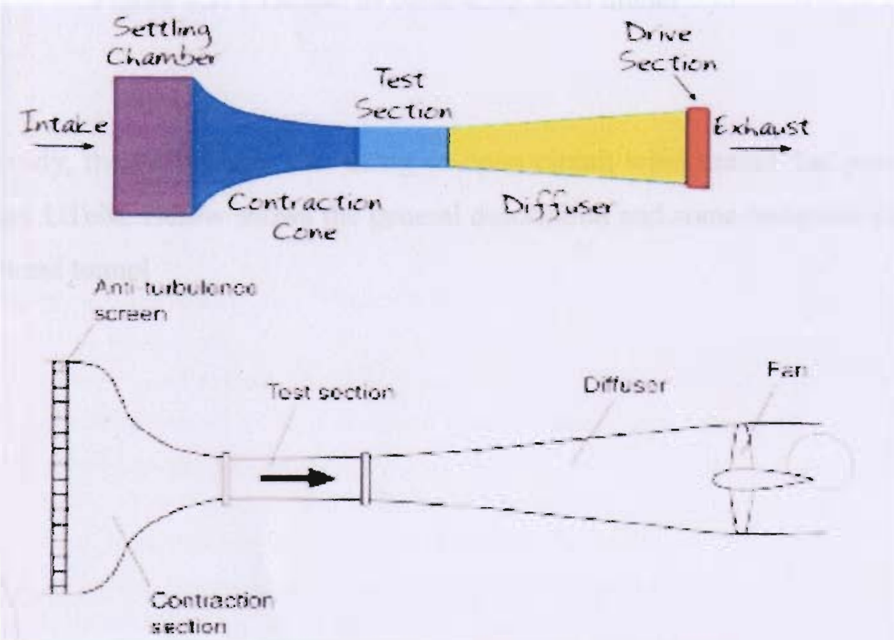

Figure 2.2 : Example of open-loop wind tunnel

2. Close-loop

   Special vanes are used to turn the air flow around the corners of the tunnel while minimizing turbulence and power loss

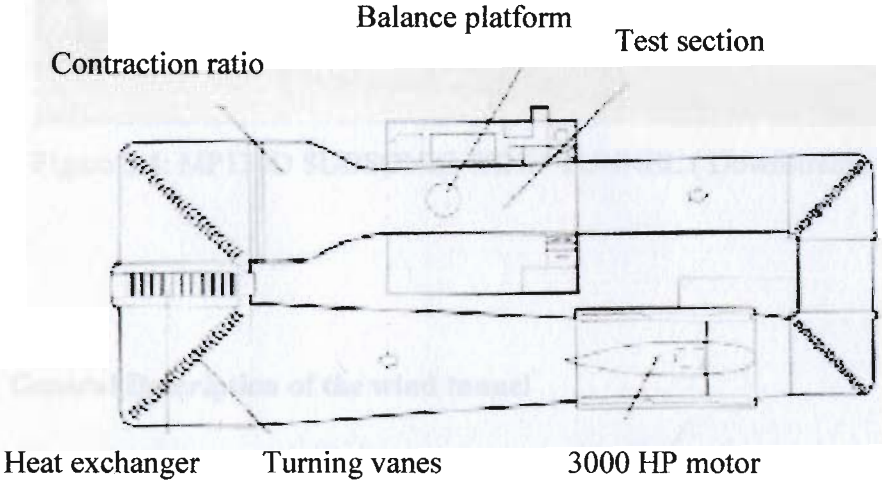

Figure 2.3: Example of close-loop wind tunnel

In this study, the experiment was using an open circuit wind tunnel that provide at the laboratory UTeM. Below shows the general description and some technical data for this type of wind tunnel

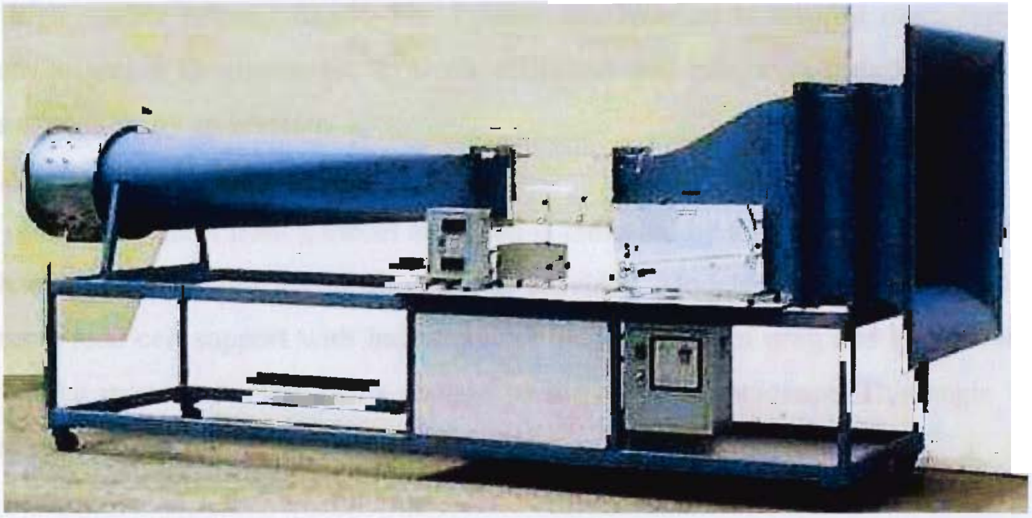

Figure 2.4: MP130D SUBSONIC WIND TUNNEL (Downstream Fan) [12]

2.3.1 General Description of the wind tunnel

This wind tunnel is an open circuit down stream fan type, design for subsonic aerodynamic studies in the experimental laboratory. Typical test capabilities are:

- Pressure and velocity measurement.
- Estimation of drag and lift coefficient of various immerged bodies, e.g. sphere, hemisphere, disc, streamlined shape, etc.
- Estimation of drag and lift coefficient of an aerofoil
- Pressure distribution around aerofoil and cylinder
- Effect of instability of a "flutter wing"
- Investigation of boundary layer around a flat plate by measuring total head distribution
- Other basic aerodynamic test

The air enters the tunnel via a flare with flow straightened, wire mesh, contraction section. The test section is transparent. Down stream of the working section is a low angle diffuser which terminates at the tunnel fan. The diffuser and contraction section have a high quality internal finish. The 7 blade fan impeller is aerofoil design cast aluminum to ensure maximum aerodynamic efficiency and minimum turbulence. Fan speed is adjustable by an inverter.

For equally spaced static pressure taps are connected to a manifold at contraction section minimize effects from a model air speed is indicated by an inclined manometer. A graph of manometer reading us air speed is provided. Models are mounted on two components load cell support with indicators for measurement of drag and lift. Model holder can be rotated to allow quick change on the angle of incidence. This angle is indicated on an angular scale at the base of the holder.

2.3.2 Technical data for MP130D SUBSONIC WIND TUNNEL [12]

- Fan : 480 mm. diameter
- Motor : 3.7 kW, 220 V/380V, 3 Ph, 50 Hz
- Inverter : 5 KVA
- Contraction area ratio : 7 : 1
- Test section : 300 mm. x 300 mm. x 400 mm
- Maximum air velocity : Over 30 m/sec
- Power supply : 220 V, 1Ph, and 50 Hz

2.4 Factors That Effect the Heat Transfer and Flow Characteristic across sphere

2.4.1 Turbulent and Laminar Flow

From scientific point of view turbulence or turbulent flow is a flow regime that characterized by velocity fluctuations and highly disordered motion. To know whether the flow is turbulent or not is to look at the flow pattern. If the pattern has burst of fluctuations in the transitional regime and then become zigzag rapidly and randomly it is a turbulent flow.

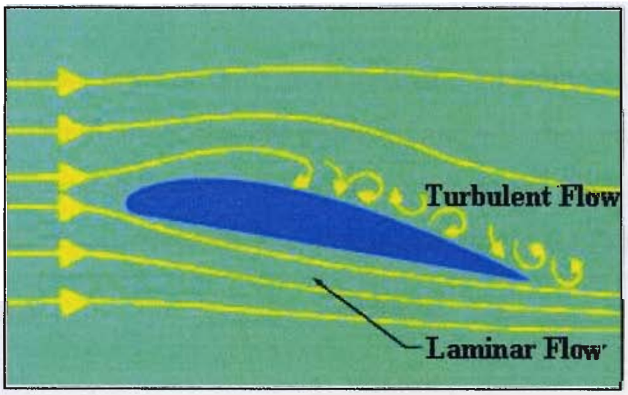

Figure 2.5 : Example of laminar and turbulence flow

There are certain factors that can results a turbulent flow which are surface roughness, high velocities and sound/mechanical waves. The Surface roughness and imperfections of a body in motion can cause the boundary layer (the layers of fluid directly in contact with the object) of the fluid in which it is moving to become turbulent more easily than if the surface were more smooth. The only factors that will include are the velocity of the fluid flow and the shape of the sphere. These two factors also can result a turbulent flow which will occur at the back of the sphere. Usually a high velocity will make a flow become a turbulent flow.

However for this study the flow is depend on the Reynolds Number and this study will describe this turbulence characteristic upstream and downstream when flowing across a sphere. Consider the flow of water over a simple smooth object, such as a sphere. At very low speeds the flow is laminar, i.e., the flow is smooth (though it may involve vortices on a large scale). As the speed increases, at some point the transition is made to turbulent flow. In turbulent flow, unsteady vortices appear on many scales and interact with each other. Drag due to boundary layer skin friction increases. The structure and location of boundary layer separation often changes, sometimes resulting in a reduction of overall drag. Because laminar-turbulent transition is governed by Reynolds number, the same transition occurs if the size of the object is gradually increased, or the viscosity of the fluid is decreased, or if the density of the fluid is increased.

2.4.2 Boundary Layer

Boundary layer is the layer of fluid in vicinity of a bounding surface . As a fluid move past an object, the fluid molecules are disturbed and will move around the object. Aerodynamic force is generated between the fluid and the object. This force depends on shape of object, speed of object, viscosity and compressibility of the fluid.

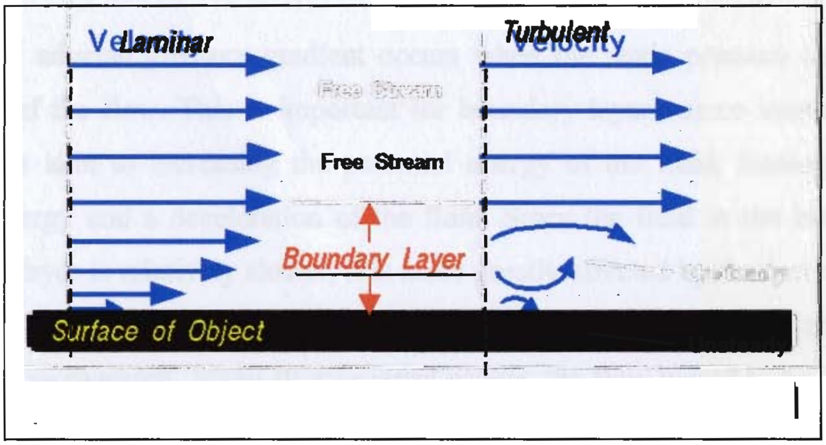

Figure 2.6 : Laminar and turbulence velocity profile

When a fluid moves past a surface, some of it will stick to the surface (usually the molecules that near the surface). So the other fluid molecules above it will slow down due to the collision between each other. This happens to all layer of the fluid until the layer that is far from the surface. The farther the molecules from the surface less collision or none will happen. This creates a thin layer of fluid near the surface in which the velocity changes from zero at the surface to the free stream value away from the surface [4]. The thin layer is the boundary layer. For flow across sphere there are actually 2 types of boundary layer involve which are the laminar boundary layer and the turbulent boundary layer ..

2.4.3 Flow Separation [14]

All solid objects travelling through a fluid (or alternatively a stationary object exposed to a moving fluid) acquire a boundary layer of fluid around them where friction between the fluid molecules and the object's rough surface occurs. Boundary layers can be either laminar or turbulent. A calculation of the Reynolds number of the local flow conditions is necessary to determine which form the flow will take.

Flow separations occurs when the boundary layer encounters a sufficiently large adverse pressure gradient. The fluid flow becomes detached from the surface of the object, and instead takes the forms of eddies and vortices.

2.4.4 Adverse Pressure Gradient [14]

An adverse pressure gradient occurs when the static pressure increases in the direction of the flow. This is important for boundary layers, since increasing the fluid pressure is akin to increasing the potential energy of the fluid, leading to a reduced kinetic energy and a deceleration of the fluid. Since the fluid in the inner part of the boundary layer is relatively slower, it is more greatly affected by the increasing pressure gradient. For a large enough pressure increase, this fluid may slow to zero velocity or even become reversed. When flow reversal occurs, the flow is said to be separated from the surface. This has very significant consequences in aerodynamics since flow separation significantly modifies the pressure distribution along the surface and hence the lift and drag characteristics.

Turbulent boundary layers tend to be able to sustain an adverse pressure gradient better than an equivalent laminar boundary layer. The more efficient mixing which occurs in a turbulent boundary layer transports kinetic energy from the edge of the boundary layer to the low momentum flow at the solid surface, often preventing the separation which would occur for a laminar boundary layer under the same conditions. This physical fact has led to a variety of schemes to actually produce turbulent boundary layers when boundary layer separation is dominant at high Reynolds numbers.

2.4.5 Wake

Wake happens at the back of the object when a fluid flows around it. Wake can be defined as the region of turbulence immediately to the rear of a solid body caused by the flow of air or water around the body. This happens because of the compression of the liquid by moving the body. It is a type of wave forms which when it spread outwards from the source, it will lose its energy. For sphere cases, wake usually happens at the back of the sphere after the separation.

2.4.6 Reynolds Number

Laminar and turbulent flows are distinguished by their behavior. The transition between the laminar flow and turbulent flow relies on pressure changes. A negative pressure gradient will delay the transition, while a positive one makes the transition occur sooner. The transition from laminar to turbulent flow depends on surface roughness, geometry, flow velocity, surface temperature and type of fluid. In fluid mechanic the factor that determines whether a flow is laminar or turbulent is a dimensionless parameter called the Reynolds Number. Reynolds Numbers is the ratio of inertial forces ($\upsilon_s \rho$) to viscous forces (μ / L) which state in equation (2.1)

$$\mathrm{Re} = \frac{\rho \upsilon_s L}{\mu} = \frac{\upsilon_s L}{\nu} \tag{2.1}$$

This is referred to as the similarity law of fluid dynamics and it indicates the ratio between viscous and dynamic forces. Experimentally, the Reynolds number has been found to tell at what point the flow becomes turbulent. If the Reynolds number is less than 2000, the flow is laminar and if it is greater than 3000, the flow is turbulent. These are just approximate values.

To classify whether the flow over a flat plate or wedge is turbulent or laminar is depend on the Reynolds Number and the correlations for the local and average convective heat transfer coefficient will be develop. But for sphere case, it is more complex. In general the flow over a cylinder may have a laminar boundary layer follow by the turbulent boundary layer and wake region depending on the Reynolds Number and diameter of the sphere as the characteristic length. The occurrence of the boundary layer transition is depends on the Reynolds Number and strongly influences the separation point position. The Reynolds Number formula as mentioned before is only for flat plate. But for this case which is a sphere the characteristic length is the diameter. So the Reynolds Number is defined as:

$$\mathrm{Re} = \frac{u_\infty d}{\nu} \qquad (2.2)$$