STUDY OF VORTEX SHEDDING AROUND BLUFF BODY USING AIR FLOW TEST RIG

RAFIUDDIN BIN AZMAN

A report submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2008

ABSTRACT

The Ahmed body represents a simplified car of 3D geometry model that can be used to study the flow characteristic in the wake of vehicles. The main objective of this study is to study the flow pattern around the model by the experimental and simulation method. The model that proposed for present work is 30^o slant angle. The experiment is conducted by using low speed wind tunnel and varied the air velocity, 0.6m/s and 6.0m/s. The time averaged results of smoke flow pattern at the curvature part is discussed. Major finding shows that the formation of small vortex is decay in time and distance as the flow travelling further. Furthermore, although overall of the smoke flow pattern results showed good agreement with previously published work, the flows structure behind the model was found to be somewhat inconsistency in a separated region over the slant and the recirculation flows behind the model. The observation indicates that the formation of small vortices over the slant is not clear apparent instead of the flows is mixing with recirculation vortices. In addition, the flow pattern of the recirculation vortices behind the model showed very similar with the previous works, indicating two vortices is formed, emphasizing in findings.

ABSTRAK

Ahmed Body mewakili geometri 3D kereta yang telah di permudahkan supaya ciri-ciri aliran dalam kawasan dimana aliran tidak stabil berlaku dapat di analisa. Objektif utama dalam kajian ini adalah untuk mengkaji aliran paten pada sekeliling model dengan menggunakan kaedah eksperimen dan simulasi. 30^{o} kecerunan pada bahagian belakang model digunakan untuk kajian ini. Eksperimen ini telah dijalankan dengan menggunakan terowong angin yg berkelajuan rendah dan dijalankan dengan halaju iaitu 0.6m/s dan 6.0m/s. Purata masa bagi aliran paten asap pada bahagian lenkungan dibincangkan.. Didapati, pembentukan *vortex* kecil terhakis pada masa dan jarak apabila aliran bergerak lebih jauh. Selain itu, walaupun keseluruhan keputusan paten aliran asap menunjukkan kesamaan dengan eksperimen sebelum in, namun struktur aliran pada belakang model menunjukkan sedikit ketidakjangkaan dalam kawasan terpisah pada bahagian cerun dan aliran berputar pada belakang model. Daripada pemerhatian menunjukkan pembentukan *vortices* kecil tidak jelas kelihatan pada kawasan cerun sebaliknya bergabung dengan putaran *vortices*. Tambahan lagi, patten aliran angin yg berputar pada bahagian belakang model menunjukkan kesamaan dengan ekspermien sebelum ini, dimana dua *vortices* terbentuk, dijelaskan dalam pemerhatian.

TABLE OF CONTENTS

CHAPTER 5 CONCLUSION

LIST OF FIGURES

LIST OF SYMBOLS

Re	Reynolds Number
V, v	Fluid velocity
D	Lateral dimension (diameter)
L	Lateral dimension (length)
υ	Kinematic viscosity of fluid
μ	Dynamic viscosity of fluid
ρ	Fluid density
θ	Diffuser angle

LIST OF ABBREVIATIONS

VIV	Vortex induced vibration
CFD	Computational fluid dynamic
CAD	Computer aided design
CNC	Computer numerical control
CR	Contraction ratio

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

A wind tunnel is a research tool developed to assist with studying the effects of air moving over or around solid objects. It is used for technical support of any major development process involving aerodynamic parts. Mostly it is used in automotive industry, laboratory research and construction. Example in automotive industry such as car, wind tunnel is used to enhance the performance of the car such as new development of windshield or front bumper for better fuel efficiency and reduce noise vibration. An advantage of using wind-tunnels is that experiments there can be performed under well controlled flow circumstances compared to experiments in the open environment. The most common experiment using wind tunnel is investigation of vortex shedding. This is because the phenomenon of regular vortex shedding behind bodies in a stream of fluid is often observed in nature. For example, vortex streets can be observed on a large scale in satellite images of cloud cover behind mountainous islands. Generally the study of vortex shedding in aerodynamics is to avoid the flow induce vibration around the body that results failure to the structure.

1.2 PROJECT PROBLEM STATEMENT

The wind tunnel is one a tool in testing the aerodynamics function that required many part with each purpose, the part are the honeycomb, screen, contraction, test section, diffuser, propeller and power source. Each part has the function, but to study the moving flow; the important part should consider is honeycomb, screen, contraction and test section. The problem of part (honeycomb and screen) is the turbulence flows intensity that enters should be lower or at least 99.50 percent of laminar flow and mean velocity is uniform. To solve the problem, honeycomb and screen design should be considered. For test section part, the cross sectional area and the length is considered and easily removable to set up model testing. The screen of the test section should be clear enough to view the flow, for example use acrylic glass. This experiment is based on existing wind tunnel that is already developed without the rig. As a result, it difficult to view the air flows at the test section. The rig or base of wind tunnel are design and developed first. For the simulation part, the number of meshing and blocks should be considering as well for better result accuracy. The boundary conditions and wall condition around the bluff model should be considered and applied corresponding to the experimental.

1.3 PROJECT OBJECTIVES

The objectives of this project are:

(i) To study the characteristics of vortex shedding around Ahmed Body

(ii) To make comparison between CFD simulation and experimental results

1.4 SCOPES OF STUDY

The scope of project covered study and analysis about characteristics of air flows enter the wind tunnel and characteristics of unsteady flow or vortex shedding. The scope of this project consists of this below:

(i) CFD simulation of air flow around Ahmed body model

(ii) Fabrication of Ahmed Body model

(iii) Fabrication of existing design wind tunnel

(iv) Experimental study of vortex shedding around bluff body

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Early nineteenth century, scientists define winds load as static load. In late nineteen century, 1879 the engineers began to study load closely. Then, the significance of dynamic response of flexible structures to the randomly fluctuating force associated with unsteadiness or turbulences that organized in wake vortex system (the term wake here is where the area of the unsteadiness or turbulence flows occurs, Figure 2.1 shows wake area). At the situation (where the wind blows aggressively), it can be more dangerous and worst for the structures if the frequency of wind is similar to the natural frequency of body or the structure (called Resonance).

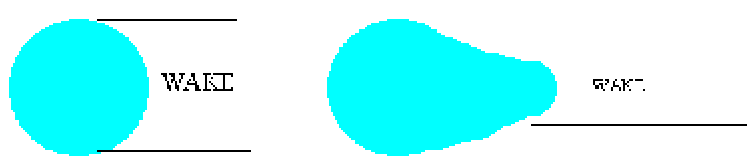


Figure 2.1: Wake occur at the back of bluff bodies

Basically, in fluid dynamic motion is to find out the effect and the characteristics of flowing fluid over a body structures. The previous studies give the major impacts on design improvement of modern structures such as bridge, aerodynamic of tall building and automotive applications. Study of unsteadiness or turbulence flow gives applications perform more efficient and better. This is because the nature force (wind) cannot be controlled.

The Ahmed reference model is a generic car-type bluff body with a slant back. It is frequently used as a benchmark test case for this kind of flow due to the flow around it retains some main features of the flow around real cars (Hinterberger et al., 2004) even it defined as simple 3D model. Furthermore, several researchers study the separation phenomena, wake structure and drag characteristics at different angles of slant back (Liu and Moser, 2003). Therein, it is shown that most of the drag of the body is due to pressure drag, which is generated at the rear end. The structure of the wake is very complex, with a separation zone and counter-rotating vortices coming of the slant side edges (Hinterberger et al., 2004). The maximum drag was found for a critical slant angle of 30º

2.2 BLUFF BODY

A bluff body is define as bodies that when subjected to a stream of fluid, suffers separation of large part of its surface or defined as bodies that are not streamline shape so that separation occurs (Williams, 2002). The occurrence of separation on a two-dimensional bluff-body flow causes the creation of two vortices in the rear region of the body (Meneghini et al., 2002).

Solution of the viscous flow (separation, shear layer and turbulence) over a bluff body has been a challenging problem in fluid mechanic for a number of years because it present in many engineering applications. Many structure, such as tall building, bridges, and offshore pipelines may considered as a bluff body member. Figure 2.2 shows the simple example of bluff body and Figure 2.3 shows PETRONAS Twin Tower’s model is one of the current examples of bluff body aerodynamic (tapered shape, combined with mild wind climate is applied to maintain the structures and keep the adverse effect of vortex excitation without using a damper system (Irwin, 2007). Figure 1.5 shows airfoil as bluff body.

Nebres and Villafranca (1992) stated that “bluff body flows may be more complicated such as flow around bridges or automobiles and becomes more complex when the body geometry or the flow field is three-dimensional”. Bluff bodies may be contrasted to streamlined bodies such as an airfoil which are shaped in such a way as to avoid regions of significant flow separation. Airfoils however becomes bluff bodies when they stall and at some operating conditions, such as an airplane wing at high angles of attack.

When a fluid flows over a body, the flow separation can cause alternating shedding of vortices from the body. Alternating vortex shedding will induces fluctuating forces and may result in structural vibrations, noise or even structural failure.

Figure 2.2: Schematic of the bluff body (Circular Cylinder) in 2-Dimensional

Source: Nebres and Villafranca, 1992

Figure 2.3: PETRONAS Twin Tower model in wind tunnel testing

Source: Irwin, 2007

Nowadays, study of bluff body is important parts in engineering applications especially on fluid dynamics and aerodynamics applications for better efficiency and performance.

2.3 VORTEX SHEDDING

A vortex or vortices is a spinning or swirl, often turbulent flow of fluid. Example of vortex or vortices will show in Figure 2.4. The speed is greatest at the center, and decrease progressively with distance from the center.

Figure 2.4: Vortex created by the passage of an aircraft wing, revealed by colored smoke

Vortex shedding is an unsteady flow that takes place in special flow velocities (according to the size and shape of the cylindrical body) as shown in Figure 2.5 and 1.6. In this flow, vortices are created at the back of the body and detach periodically from either side of the body and formed boundary layer on the surface of the body. Vortex shedding also produces shears layers from this boundary layer. At a certain instant, the growing vortex draws the opposing shear layer across the near wake. The approach of oppositely signed vorticity, cuts off further supply of circulation to the growing vortex, which is then shed and moves downstream (Meneghini et al., 2002).

This vortex also will induce vibration on the body called vortex induced vibration (VIV) and become unstable due to natural disturbances in flow. Vortex Induced Vibration (VIV) are motions induced on bodies facing an external flow by periodical irregularities on this flow. VIV occurs when shedding vortices exert oscillatory forces on a cylinder in the direction perpendicular to both the flow and the structure (Gabbai and Benaroya, 2004).

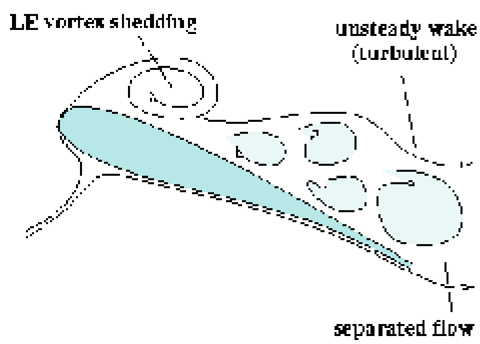


Figure 2.5: Vortex shedding behind the body (at high angle of attack)

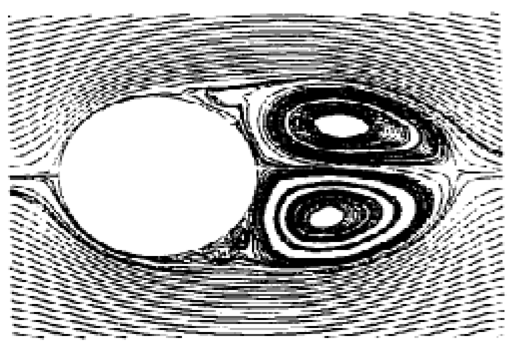


Figure 2.6: Vortex shedding at the back of body. Streamline pattern in the cylinder wake of impulsively started flow for Re = 500 and t* = 3.0s

Source: Akbari, 1999

Generally, most common study of vortex shedding on bluff bodies is von Kármán Vortex Street, Vortex Induced Vibration, vortex excitation and the characteristic of vortex shedding.

2.3.1 Von kármán vortex street

A Von Kármán Vortex Street is a repeating pattern of swirling vortices caused by the unsteady separation of flow over bluff bodies. The Von Kármán vortex street is a typical fluid dynamics example of natural instabilities in transition from laminar to turbulent flow conditions, as shown in Figure 2.7.

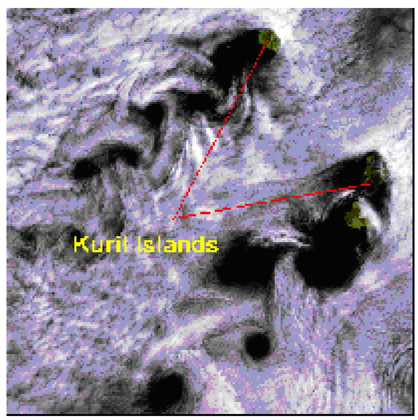


Figure 2.7: Von Karman vortices formed over the islands of Broutona, Chirpoy, and Brat Chirpoyev, Kuril Islands

A vortex street will only be observed over a given range of Reynolds numbers, typically above a limiting Re value of about 90. In the wake of the cylinder, when the flow reaches a Reynolds number value between 40 and 200, alternated vortices are emitted from the edges behind the cylinder and dissipated slowly along the wake, see Figure 2.8.

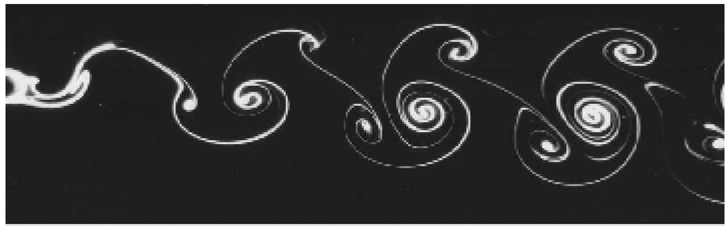


Figure 2.8: Von Kármán Vortex Street, wake pattern behind an oscillating cylinder at Re = 140

Source: Aref et al., 2006

Theodore von Kármán is the person who first stimulated widespread interest and published the first theoretical study of vortex streets in 1911. (Schatzman, 1981), who studied the analysis of a model for the Von Kármán Vortex Street found that the linear stability of the point vortex has been generalized to vortices of finite size and can stabilize the array.

2.4 REYNOLDS NUMBER EFFECT

The vortex shedding from a bluff body is a function of the Reynolds Number (Re). The flow characteristics of wind passing across bluff body are depend on magnitude of inertial to viscous within the flow (the parameter are called Reynolds Number). The Reynolds Number is defined as:

$$\mathrm{Re} = \frac{VD}{\upsilon} \quad \text{or} \quad \mathrm{Re} = \frac{\rho VL}{\mu} \tag{2}$$

Where V is the wind velocity, D, L is the lateral dimension of the body, υ is kinematics viscosity of air, μ dynamic viscosity of air and ρ is density of air.

2.4.1 Formation of vortex shedding

Different Re will effect the formation of vortex shedding over bluff bodies. For example, the Figure 2.9 below shows the formation of vortex shedding (circular cylinder) by varied the Reynolds Number value, no separation occur at Re<5. For 5<Re<40, separation occurs and two symmetric fixed eddies (vortex) are formed in the wake on two side of the cylinder. For 40<Re<190 the wake is laminar. The wake is turbulent and remain turbulent at Re>300. For Re above 3.5×10^6, the boundary layer is fully turbulent or unsteadiness (Akbari, 1999).

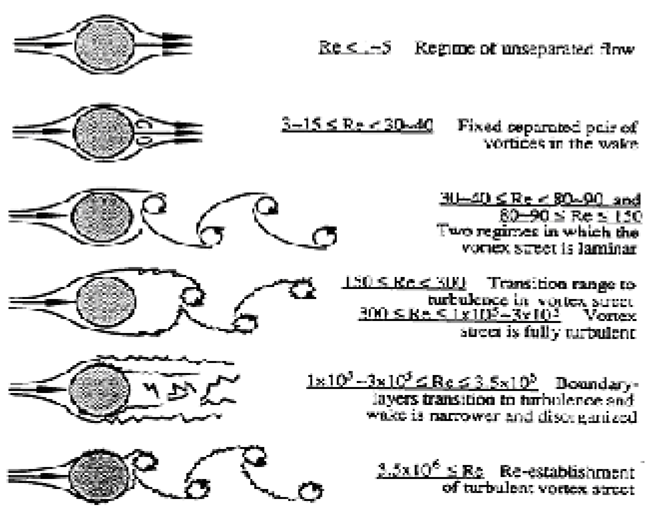


Figure 2.9: Regimes of flow around circular cylinder

Source: Nebres and Villafranca, 1992

2.4.2 Drag characteristics

The overall flow structure and the location of separations will not change significantly with tunnel speed, mainly due to the geometry model having sharp edges at the rear end (Vino et al., 2004). The work Spohn and Gillieron (2002), which was conducted in a water tunnel at a Reynolds number of $3x10^4$, also suggests that the

Ahmed model exhibits small changes in flow characteristics as a function of Reynolds number (Vino et al., 2004). Thus, it is apparent that the drag characteristics are affected by changes in Reynolds number. Drag is at its highest at very low speeds and gradually decreases with increasing Reynolds number. Figure 2.10 shows the effect of drag characteristic corresponding with Reynolds Number.

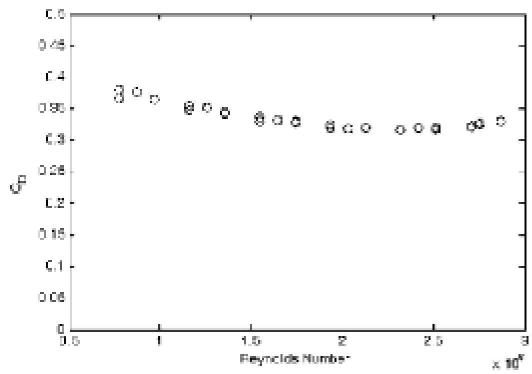


Figure 2.10: Reynolds number effects on Ahmed model geometry

Source: Vino et al., 2004

2.5 AHMED BODY THEOREM

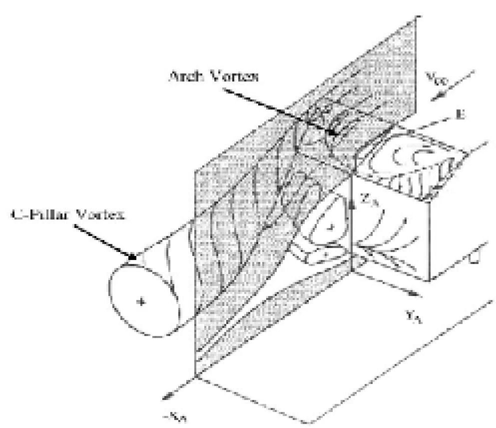


Figure 2.11: Proposed vortex system for hatchbacks, Ahmed et al. 1984

Source: Vino et al. 2004

Ahmed et al. observed that for a base slant angle less than 30^{o} (critical angle or maximum drag is observed), a separating shear layer turns up from the sides of the rear slanted edge (or Counter rotating vortices) and rolls into two longitudinal vortices. Also, the flow separates from the roof and then reattaches on the slant near the vertical base, forming an arch-shaped separation vortices over the surface (Vino et al., 2004). Upon leaving the rear of the backlight, the flow again separates from the top and bottom edges of the vertical base, forming two separation bubbles, one above the other and in opposing directions. Figure 2.11 shows the proposed of vortex system.

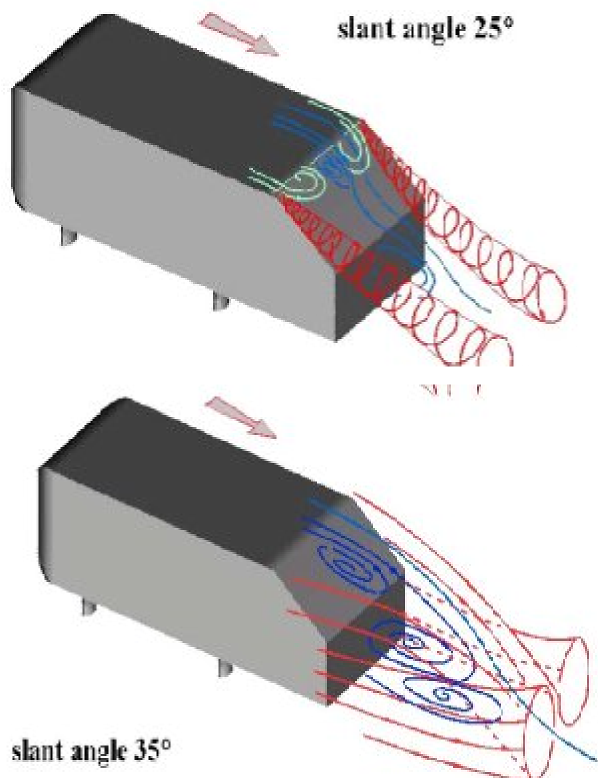


Figure 2.12: Development of flow in rear of Ahmed Body model (from Ahmed et al. 1984)

Source: Rodi, 2004

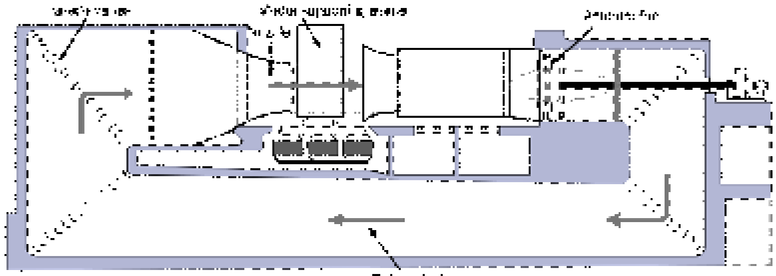
Figure 2.12 shows the development of flow at rear part in 25^o and 35^o slant angle. Ahmed proposed that below 30^o slant angle, strong counter-rotating vortices are observed from the sloping edges of the body and separates in the middle region before reattaches at the sloping surface (Rodi, 2004). Above this angle, the counter-rotating vortices are much weaker, the separation occurs along the entire top and the side edges, and there is no reattachment on the sloping surface, a sudden drop in drag occurs which corresponds to a drastic change in the flow in the wake (Rodi, 2004). The differences are illustrated on Figure 2.12.

2.6 EXPERIMENTAL METHOD

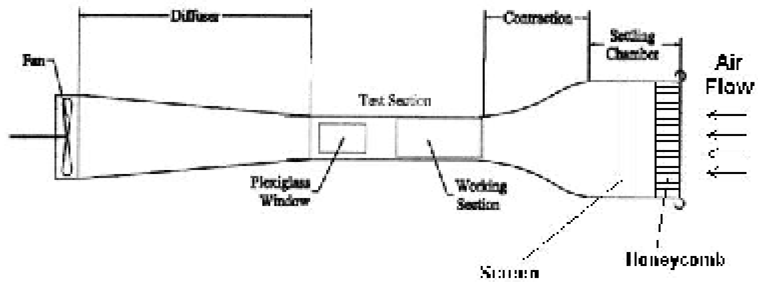
There are several methods to investigate the vortex shedding around bluff bodies such as using numerical method, using simulation software (known as Computational Fluid Dynamics, CFD) or using air flow testing rig.

2.6.1 Air flow test rig

Experiment of vortex shedding using air flow testing rig can be done by using wind tunnel. Wind tunnel may be classified according to their basic architecture (open loop, closed loop) in Figure 2.13, according to the speed (subsonic, transonic, supersonic, hypersonic), according to the air pressure (atmospheric, variable-density), sizes (small, full-scale) or air flow (blower, suction). Both types then are categorized in subsonic, transonic, supersonic and hypersonic wind tunnel.



(a) Close loop



(b) Open loop

Figure 2.13: Type of wind tunnel

The following is part of open circuit of wind tunnel and functions of the parts.

1. **Settling Chamber** (contain Honeycomb and Screen)

   - **Honeycomb** - Honeycomb is installed to straighten the flow as well as to attenuate some high frequency noise. Honeycomb removes swirl from the incoming flow and minimizes the lateral variations in mean velocity and the yaw angles are less than 10^o. Honeycomb comes in different shapes including circular, hexagonal, rectangular and etc. Below shows example oh honeycomb shapes.

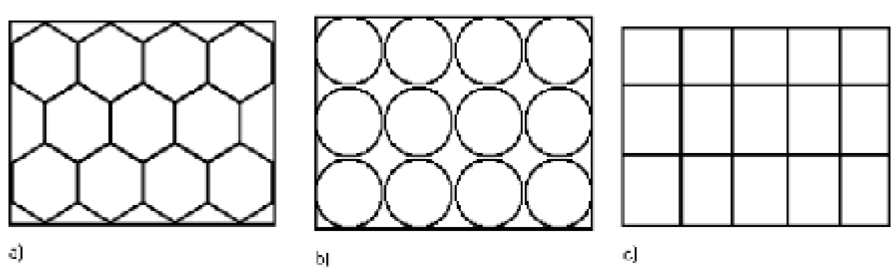


Figure 2.14: Type of honeycomb