A COUPLED SIMULATION METHOD FOR A VORTEX-INDUCED VIBRATION SYSTEM

MONG GUO REN

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

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2017

DECLARATION

I declare that this project report entitled "A Coupled Simulation Method For A Vortex-Induced Vibration System" is the result of my own work except as cited in the references

Signature : ....

Name : ....

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 (Thermal-Fluids).

Signature :. ....

Name of Supervisor : ....

Date : ....

DEDICATION

To my beloved father and mother.

ABSTRACT

Fluid structure interaction is a phenomenon that happened around us every day. Whenever there is fluid flowing around a stationery object, fluid induced effect will occur on the object. Simulation carried out using computer aided engineering (CAE) usually does not include body motion to reduce the simulation time. This project is carried out to identify will the body motion effect on flow behaviour in high Reynolds Number. Airfoil cross-section of NACA0018 is chosen to be simulated using ANSYS Fluent v16. Fixed airfoil simulation is carried out to find the presence of vortex shedding and Strouhal Number. Two more categories of simulation is carried out using Reynolds Number of 10^5 and 10^4. Both simulations have a system with different stiffness and natural frequency. Result shows that the Strouhal Number of vortex shed remains at 0.732 despite a change in system stiffness and Reynolds Number

ABSTRAK

Cecair interaksi struktur adalah satu fenomena yang berlaku di sekeliling kita setiap hari. Apabila terdapat cecair mengalir di sekitar objek, objek akan mengalami sejenis kesan daripada aliran cecair. Kebiasaannya simulasi yang dijalankan menggunakan komputer kejuruteraan (CAE) tidak termasuk pergerakan badan untuk mengurangkan masa penyelakuan. Projek ini dijalankan untuk mengenal pasti adakah pergerakan badan menjejaskan tindak balas badan yang disebabkan oleh aliran bendalir dalam tinggi Nombor Reynolds. Keratan rentas aerofoil NACA0018 dipilih untuk disimulasikan menggunakan ANSYS Fluent v16. Simulasi aerofoil dijalankan untuk memerhati tindakbalas aerofoil dalam aliran angin. Dua kategori simulasi dijalankan menggunakan Reynolds Number 10^5 & 10^4. *Sistem yang mempunyai kekukuhan yang berbeza digunakan. Keputusan dari simulasi menunjukkan bahawa Nombor Strouhal kekal pada 0.732 walapun perubahan dalam kekukuhan system dan Nombor Reynolds.*

ACKNOWLEDGEMENT

Firstly I would like to thank God for giving me this opportunity to study in UTeM and having this chance to complete my Final Year Project.

Dr. Cheng See Yuan who is my supervisor deserve my gratitude and appreciation. Thank him for giving me this opportunity to carry out my final year project research with him. His guidance throughout the whole year and encouragement when I needed it most has supported me both physically and mentally.

Secondly, I would like to thank my family members that gave me mental support until the completion of this project. My friends, coursemates who have been with me through thick and thins makes me who am I today. They too deserve my gratitude.

Last but not least, I want to thank UTeM for equipping me with knowledge, providing instruments for us to learn more efficiently and all the lecturers who have taught me since the time I stepped into UTeM. Without all of that, I wouldn‟t probably be able to complete my Final Year Project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

CHAPTER 1

INTRODUCTION

1.1 Background

An object subjected to flowing fluid will experience a phenomenon known as fluid structure interaction. This interaction will generate resultant force that is resolved into the lift and drag directions for convenient in the analysis.. When fluid encounter a blunt object, wake region will present behind the body where back flow takes place. One types of wake is in vortices form. This phenomenon is due to separation of boundary layer that bounds the wake and free stream which leads to fluid rotation. These rotating fluid result in formation of individual vortex which sheds at the rear end of the body and travel down the wake. Typically, an vortex shedding which is periodic and in alternating form will take place in the wake field, this is called a vortex street.

At some instances, the vortex shedding has a pattern which is not symmetrical about the body. There are some vortices being shed on the top surface of the body and some are at the bottom of the surface. This will cause irregular pressure distribution on the body hence causing lift force at both sides of the body to be exerted periodically. This occurrence is known as vortices induced vibration (VIV). This periodic shedding action has its own frequency. When the body‟s natural frequency matches with the frequency of this oscillation, „Lock-in‟ or also known as resonance will happen. This will amplifies the body amplitude of vibration creating displacement on the object.

A fixed body experiencing fluid induced vibration will has a different wake region when compared to an elastically mounted body which is allows to moves. These elastically mounted bodies will experience a motion which is in translation or rotation or both. This means that the body will moves whenever there is fluid induced effect on the body. At this moment the wake region of the body might be affected as the moving body will cause a change in the body orientation. When the wake region changes it will affect lift and drag forces of the body in the flow field. Hence it is important to know whether the body motion affect flow behaviour when compared to a fixed body where the wake is constant throughout.

1.2 Problem Statement

There are numerous objects of various shapes that experience fluid flow for example house, bridge, fan, trees etc. Fluid structure interaction is the phenomena that occurs when a flowing fluid encounter an object. During this interaction, the objects will response in a particular way due to the flowing fluid. This is known as the fluid induced effect. In many simulations, the response of body which sometimes comes in translational and rotational manner or even deformed object is neglected. This reduces the simulation time and simplified the simulation case. However in real life situation, this response of the body is present and might cause changes to the properties or performance of the system.

Different cross-sectional object will react differently. In this project an airfoil cross-section shape is chosen to be investigated. This is because there are lots of airfoil applications around us for example the spoiler of a car, the wing of an airplane, blade of a helicopter, blade of fan, blade of wind turbine and blade of compressor rotor.

The vortex induced vibration on the airfoil will be investigated in this project. A fixed airfoil and an elastically mounted airfoil will be setup. Changes in vortex shedding frequency, magnitude of lift force and magnitude of airfoil movement along the y-axis will be determine by manipulating the stiffness of the system and the fluid‟s Reynolds Number.

1.3 Objectives

The objectives of this project are as follows:

- To identify flow behaviour in fix body simulation and moving (1DOF) rigid body simulation
- To examine impact of body motion on flow characteristics
- To identify the vortex shedding in wake region of airfoil.
- To observe the effect of aerodynamic forces on an elastically mounted airfoil

1.4 Scopes Of Project

The scopes of this project are:

- Fixed airfoil simulation to identify vortex shedding
- Simulation of elastically mounted (1DOF) airfoil (rigid body) with different stiffness
- Compare the changes in airfoil response in fix and elastically mounted simulation

CHAPTER 2

LITERATURE REVIEW

2.1 Fluid Structure Interaction (FSI)

Fluid Structure Interaction is the phenomenon that occurs at the interphase between an object and the surrounding environment fluid. Whenever fluid flow passes a stationary object or a moving object moving pass a stationary fluid domain, there will be effect imposed by both the fluid and object on each other. Happenings of FSI which can be seen at for instance oil-rig risers and bridge (Zahari & Dol, 2014) and in medical tools (Hessenthaler et al., 2016) involve the coupling of unsteady fluid flow and structure motion. The mechanism that causes elastic structure to vibrates and turn to self-excite while submerge in a flowing fluid is hard to estimate and predict due to mechanical properties of structures and properties of the fluid itself.

There are experiments carried out to investigate flowing fluid relationship with the structure it is impose on, result shows that they are both highly interrelated elements. In Chee Chew Wong articles, (2011) metal plates in laminar flow is studied and the transient response of the plates under FSI conditioned is determined. Validation for FSI cases through real time investigation and experiments is vital to prove the simulations obtained from Computer Aided Engineering Tools are trustworthy. Lienhart & Pereira Gomes, (2006) uses Particle Image Velocimetry to measure flow field and structure deflection in validating their results. A two dimensional reference structure of a cylinder is immersed in laminar and turbulent flow to examine their interaction behaviour. Therefore simple structure such as cylinder or plate to complex structure such as bridge (Hansen et al., 2015) which experience FSI are among the topic of interest of structural engineers as they provides useful and relatable information to the objects in our daily lives.

2.1.1 Vortices Induced Vibration (VIV)

Vortices Induced Vibration (VIV) is a typed of FSI that happens on a structure. This is an effect of vortices shedding in the wake flowing through bluff body structure such as column, risers and mooring lines. Structure Hydrodynamic loading will be affected by change in vortices which will cause structure to vibrate (Nguyen & Nguyen, 2016). VIV occurs when pressure on surface fluctuates; this leads to fluctuates of lift force thus producing cross stream oscillation. These oscillations sometimes can be large and sometimes is too small to notice depends on the wake behind the object. Gharib, (1999) experiments using cylinder of various mass ratio states that the vortex wake tends to be almost identical regardless of the geometry being simulated. This also means that object subjected to fluid flow will experience either little or a large amount of vortex wake where fluid separation occur at the rear end of the body. There is presence of vortex shedding behind the race car clean wing and a shear layer between regions of positive and negative vorticity (Kuya, et al, 2009). This confirms that although the wind is aerodynamic in size, there is also vortex formation which is due to wake region.

Hansen, (2007) carried out an experiment in wind tunnel using geometry with different cross-section. Among the tested geometries are circular cylinder, sharp-edged sections, octagons and bridge decks. From the result of Hansen, (2007), significant vibration level will causes displacement of structure when the wind reaches slightly above 6m/s. This shows activity of VIV on these structures with different cross-section caused by the flowing wind.

Besides of VIV which is directly related to the FSI, there is also Wake Induced Vibration (WIV) which also has a similar effect when compared to VIV. The difference is that WIV happened only when there is wake region that contains sufficiently large pressure differences or also can be known as Karman Vortex Street.

2.1.1.1 Factors Affecting VIV

In Pantazopoulos & Pantazopoulos findings, (1994), they state that important parameters in VIV are lift coefficient, shedding frequency (Strouhal number), correlation length and shedding frequency bandwidth. Hansen et al., (2015) mention VIV measured in dynamic test rig depends strongly on Reynolds Number. Bluff bodies surface will always experience region of vorticity where flow is more or less coherent. Figure 2.1 (Hansen et al., 2015) provides clear visualization on the flow separation region on bluff body where back flow occur which leads to formation of small votrices in the shear layer region near the boundary of the object. On sharp-edge body separation flow defined by location of edges while rounded bodies depends on air velocity which is also directly related to Reynolds number.

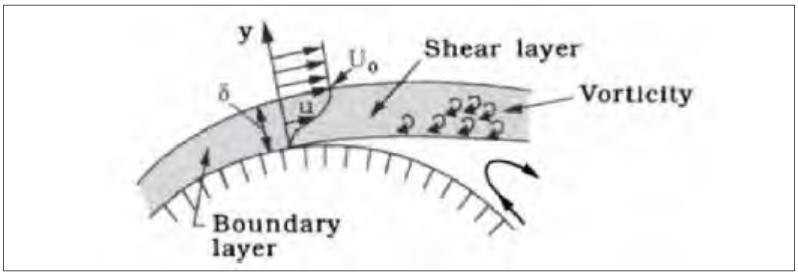

Figure 2.1: Detailed Sketch Of Flow Near Separation (Hansen et al., 2015)

Hansen et al., (2015) uses different configuration of streamlined single box grinder to investigate the effect of VIV. It is found that Strouhal number increase with Reynolds number and the resonance wind velocity for VIV is governed by Strouhal number.

During vortex formation, when the vortices are not produced symmetrically with respect to the mid-plane of the body, different lift forces formed at each side of the body leading to motion tranverse to the flow. Strouhal Number which is an important term in vortex shedding is shown in equation below.

$$S = \frac{f_v D_c}{V} \tag{2.1}$$

Where f_v is the vortex shedding frequency at rest, D_c is the diameter of circular cylinder and V is the velocity of ambient flow. Frequency of vortex shedding is related to Reynolds number, Re which is also linked to flow velocity, viscosity of fluid and characteristic length of object (Abhiroop Jayanthi, 2008).

A new parameter termed „effective stiffness" by Gharib, (1999) will also affect the chances of VIV. By reducing the structure to fluid mass ratio, lock-in behaviour or resonance can also be prevented. However it is also found out that at low mass ratio, VIV can occur with or without lock-in behaviour. Self-excite oscillation and forced oscillations are the types of VIV where self-excited oscillation occurs naturally and forced oscillation can be controlled independently of fluid velocity.

2.1.1.2 Wake Region in VIV

Williamson & Govardhan, (2004) uses an elastically mounted cylinder as an example for VIV system. Certain wake pattern is induced by body motion for example the 2P, 2S, P, C and S mode. These are all due to imbalance pressure distribution across the surface of the body of cylinder when fluid flows pass it.

An object experiencing fluid flow will experience two effects which is drag force and lift force. When an object subjects to VIV, fluctuating lift force can be seen on a body. When the pressure of both upper and lower side of the object fluctuates with time, the lift will be affected too. These are dimensionless coefficients defined by:

$$C_L = \frac{F_L}{\frac{1}{2} p V^2 A_L} \tag{2.2}$$

$$C_D = \frac{F_D}{\frac{1}{2}pV^2A_D} \tag{2.3}$$

Where F_L and F_D are the lift and drag forces, p is the density of the fluid and V is the velocity. A_L and A_D are reference area. The size of the wake region will determine the magnitude of these two forces acting on the body(Lienhard, 1966).

2.2 Airfoil

Airfoil shape is used in car spoiler and wing, this shape is usually inverted to provide desired effects. Inverted airfoil has a low pressure suction region on the lower surface and high pressure suction region on the higher surface of airfoil geometry. Airfoil geometry can be summarize by few parameters for example are maximum thickness, maximum chamber, position of max thickness and many more. Ahzalilov, (1996) found out that shape of pressure distribution relates closely to airfoil performance as shown in Figure 2.2. It is shown that the geometry or design of the cross-section will cause the pressure distribution across the airfoil surface to change. Holmes, (2008) state that large wing surface area creates more lift while the increase of aspect ratio will induce less air resistance. Breu,et al, (2008) also mention that the drag reduction of a vehicle by implementing a rear spoiler is strongly dependent on the shape of spoiler.

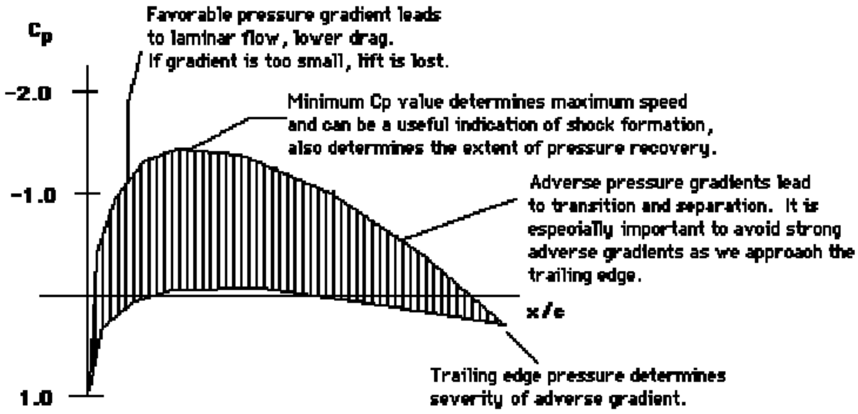

Figure 2.2: Relationship of Airfoil Performance and Its Dimension (Ahzalilov, 1996)

Since most aerofoil profile is used in airplane to create a large lift while taking off, by inverting the shape it will create a sufficient negative lift force also known as down force on the vehicle. Stiffness of system will affect the behaviour of the aerofoil which the dynamic response of the aerofoil is dependent on the location of pitching axis says (Z.Peng, 2009). Experiment on spoiler effect on wake region was carried out by et al, (2011) and its found out that the deflection of spoiler enlarge the wake region by 0.79%. Tsai, et al, (2009) also found out that by installing spoiler, lift coefficient can be reduced which leads to improved high speed driving condition and vertical stability of driving. From the experiment, when spoiler is placed on the trunk top, drag coefficient increase while lift coefficient decreases. Trailing vortex that is similar to that of a flying plane is produce due to pressure difference on the upper and lower surface of spoiler. This phenomenon also occurs in the work of Meederira, (2014) which saw vortex generation from end plates which can be seen in Figure 2.3. This shows that fluid flowing pass a airfoil profile will generate vortex in the wake region.

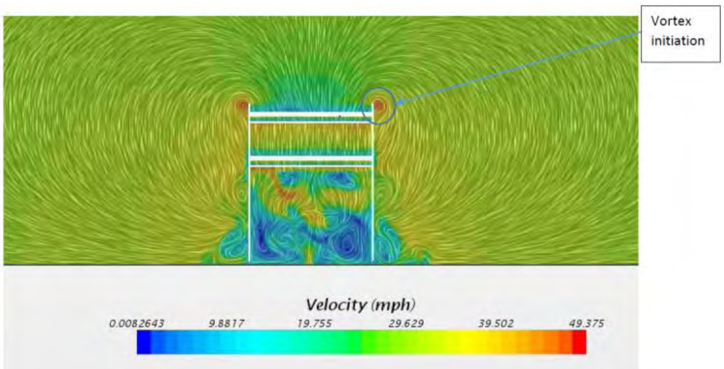

Figure 2.3Vortex Generation From End Plates (Meederira, 2014)

Meederira, (2014) result indicates that when the AOA is increased, separation region behind the body become more obvious. This separation flow will affect the lift and drag force induced on the body. Kieffer, et al, (2006) and Meederira, (2014) found out that hydrodynamic performance of foils will be affected by the Angle of Attack (AOA) of the foils. Holmes, (2008) state the coefficient of lift increases with increasing angle of attack however at some angle flow separation happens and enlarge which will reduce the lift coefficient. Huang (2011) states airfoil when placed at post-stall angle of attack might response as if a bluff body. Flow separation problem can be tackle through element of high aspect ratio. Other than that, endplates of the rear wing in 3D simulation avoid trailing vortices formation which is due to air leakage in wing tip. Experiments carried out by Elsayed et al., (2011) found out the fixing spoiler on aircraft wings is able to accelerate vortex breakdown in the wake region. By deflection spoiler by 10° peak vorticity value is reduced to 0.46.

2.3 Mass Spring Damper System

Mass spring damper system is used in many experiments especially those that involve object"s motion. Equation that govern the object in this system is $m\ddot{x} + b\dot{x} + kx = W$ shown in Figure 2.4. Zhang & Ji, (2016) uses an elastically mounted 2DOF system that include an airfoil and circular shape.

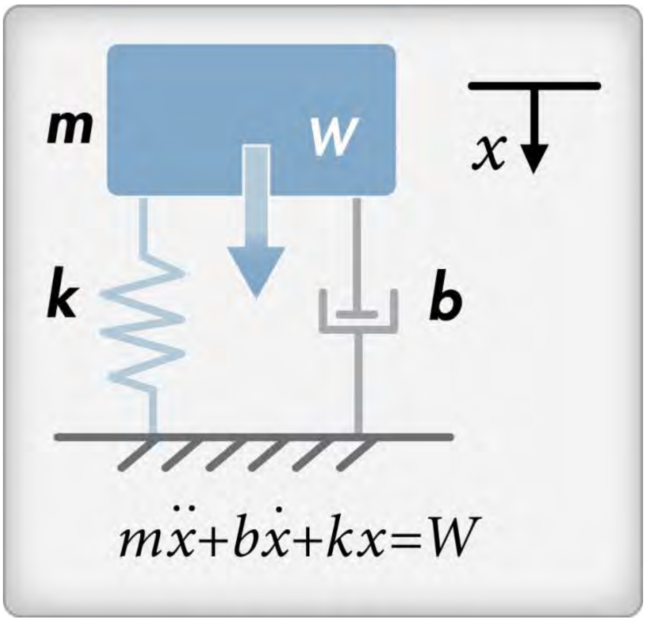

Figure 2.4 Mass-Spring-Damper System

Motion of object is differentiate into 6 types, translation in x, y & z axis and rotational motion in x, y & z axis.

2.3.1 Natural Frequency

The natural frequency of a system is the frequency where system oscillates when it is acted by an external force. The natural frequency of a system is calculated by,

$$F_N = \frac{1}{2\pi}\sqrt{\frac{k}{m}} \tag{2.4}$$

Resonance or lock-in occurs when the natural frequency of the structure is in sync with the frequency of an outside force. During this period, exceptionally strong response will be obtained from the structure causing the structure to vibrate or moves with great amplitude. In Derakhshandeh, et al, (2016) free-decay tests are conducted to obtain the natural frequency of the fundamental bending mode of the cantilevered aerofoil. From the dynamic response of the subject, 4.70Hz is the natural frequency of the cantilevered beam which is away from the frequency of oscillation hence structure does not experience resonance effect.

For the unsteady pressure at the wake vortices to induce the body to response, Williamson & Govardhan, (2004) found out that the vortex formation frequency has to be closed enough to the body natural frequency. Besides from this finding, it is also found out that bodies can conceivably vibrates at high amplitude at hundreds of times the natural frequency. Sufficient large body movement during this phenomena may affect the wake flow according to Gharib, (1999). The vortex growth period and phase can be affected resulting in stronger vortices with altered patterns. This will cause an increase in spanwise correlation of the wake, drag and lift. Results found that 40% of vortex shedding frequency can be shifted and this effect are not only limited to circular cylinders.

CHAPTER 3

METHODOLOGY

Circle shape is the most common cross-section used in Vortex Induced Vibration study such as by Singh & Mittal, (2005) and Mysa, et al, (2016). The shape or cross-section of the object to be studied needed to be determined. Review from previous study found out that there are lots of researches on airfoil profile in laminar flow but there are not many studies on turbulent flow. Hence this work will focus on turbulent flow across airfoil cross-section shape. The shape of airfoil comes in many while most profiles are based on NACA number. A symmetry profile of NACA0018 which has been investigate its properties by Nakano, et al, (2007) is chosen to be investigated.

First and foremost the formation of vortex is to be observed behind the airfoil. This is done to show vortex shedding behind airfoil profile as found by Meederira, (2014). If vortex shedding is not observed, the angle of attack, AOA of the airfoil is adjusted and re-run the simulation. After obtaining steady vortex shedding, fast fourier transform, FFT is done on the oscillating lift data to find its peak frequency which is the vortex shedding frequency.

The next part of this project will be carried out by allowing translation motion of the rigid airfoil body. A user defined function, UDF which uses 6 degree of freedom, 6DOF is applied to the rigid body. The feasibility of this UDF is validated by running a simulation using a circle as our object which is carried out by Singh & Mittal, (2005). Simulation with a non-dimensional mass of 10, reduced velocity of 4.6 and Reynolds Number of 100 is carried out by allowing transverse and in-line motion of circle.