ANALYSIS OF VORTEX SHEDDING IN A VARIOUS BODY SHAPES

NAZIHAH BINTI MOHD NOOR

A thesis submitted in fulfillment of the requirement for the award of the
Master of Mechanical Engineering

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

JULY 2015

Dedicated with much love and affection to my beloved parents, who inspired me and sparked my interest in pursuing higher education, who are praying for me and provided me with support, help and encouragement every moment while studying Master of Mechanical Engineering that I have followed.

To my lovely husband, Kamarul Hasnan.
You are the love of my life, my strength and support.

To my son Muhammad Fairuddin and my daughter Anis Saffiyah.
You are great kids and brighten me every day.

ACKNOWLEDGEMENT

All praises to Allah Almighty who created the universe and knows whatever is in it, hidden or evident, and who bestowed upon me the intellectual ability and wisdom to search for its secrets.

The production of this thesis involves many valued contributors from a number person. First and foremost, praises and thanks to My Lord Allah, The Almighty, The Most Merciful and The Most Graceful. Alhamdulilillah, finally I managed to complete this thesis by the praise of Allah and His guidance and blessings.

This project would not have been possible without the support of many people. I would like to express a very special gratitude and thankfulness to my supervisor Dr. Izzuddin Bin Zaman @ Bujang who is giving me valuable taught, encouragement and guidance that lead me in fulfill my master project. Indeed, all knowledge, qualification and advice are useful and very much appreciated.

My special appreciation is due to classmates and lectures for their paramount patience and tolerance in guidance me for this development.

Lastly, to my husband, my kids and my parents, I would like to forward my obliged to them for their continuous support during my study period, their patience and benevolence.

Thank you all from the bottom of my heart.

ABSTRACT

The study of various bluff body shapes is important in order to identify the suitable shape of bluff body that can be used for vortex flow meter applications. Numerical simulations of bluff body shapes such as circular, rectangular and equilateral triangular have been carried out to understand the phenomenon of vortex shedding. The simulation applied k-ε RNG model to predict the drag coefficient of circular in order to get a closer result to the previous research. By using CFD simulation, the computation was carried out to evaluate the flow characteristics such as pressure loss, drag force, lift force and flow velocity for different bluff body shapes which used time independent test (transient) and tested at different Reynolds number ranging from 5000 to 20000 with uniform velocities of 0.675m/s, 1.35m/s, 2.065m/s and 2.7m/s. It is observed that triangular shape gives a low coefficient of pressure loss with value 1.7538 and the low coefficient of drag coefficient is rectangular shapes correspond to 0.7613. By adding splitter plate behind the bluff body, it can reduce drag force and pressure loss. A triangular with splitter plate give a highest percentage of pressure loss and drag force with 29% from 0.1969 to 0.1397 for pressure loss and 1.746 to 1.2515 for drag force. The result concludes that in order to get better performance, vortex flow meter requires bluff body with sharp corner to generate stable vortex shedding frequency.

ABSTRAK

Kajian terhadap pelbagai bentuk badan pembohongan untuk mengenal pasti bentuk badan pembohongan yang sesuai untuk digunakan pada aplikasi meter aliran pusaran. Simulasi berangka yang dilakukan pada badan pembohongan adalah berentuk bulat, segi empat dan segi tiga sama sisi untuk memahami fenomena gugusan pusaran. Penggunaan simulasi k-ε model RNG digunakan bagi meramal nilai pekali seretan pada bentuk bulat untuk mendapatkan nilai yang lebih hampir kepada kajian lepas. Dengan menggunakan simulasi CFD, pengiraan telah dilakukan untuk menilai ciri-ciri aliran seperti kehilangan tekanan, daya seret, daya angkat dan halaju aliran untuk bentuk badan pembohongan yang berbeza dengan menggunakan ujian masa bebas dan dan diuji dengan nombor Reynolds dari 5000 hingga 20000 dengan halaju seragam iaitu 0.675m/s, 1.35m/s, 2.065m/s dan 2.7m/s. Dapat diperhatikan bahawa bentuk segi tiga memberikan pekali yang rendah bagi kehilangan tekanan dengan nilai 1.7538 dan pekali yang rendah bagi pekali seretan adalah berbentuk segiempat dengan nilai 0.7613. Dengan menambah plat splitter di belakang badan pembohongan, ia boleh mengurangkan daya seret dan kehilangan tekanan. Segitiga dengan plat splitter memberikan peratusan tertinggi kehilangan tekanan dan daya seretan dengan 29% daripada 0.1969 kepada 0.1397 untuk kehilangan tekanan dan 1.746 kepada 1.2515 untuk daya seretan. Secara keseluruhannya, bagi mendapatkan prestasi yang lebih baik, meter aliran vorteks memerlukan badan pembohongan yang bersudut tajam untuk menjana frekuensi vorteks yang stabil.

TABLE OF CONTENT

	TITLE		i
	DECLARATION		ii
	DEDICATION		iii
	ACKNOWLEDGEMENT		iv
	ABSTRACT		v
	ABSTRAK		vi
	TABLE OF CONTENT		vii
	LIST OF FIGURES		x
	LIST OF TABLES		xiii
	LIST OF SYMBOL		xiv
	LIST OF APPENDICES		xv
CHAPTER 1	**INTRODUCTION**		**1**
	1.1	Research Background	1
	1.2	Problem Statement	2
	1.3	Objective of Study	3
	1.4	Research Scope	3
	1.5	Significant of Study	4
CHAPTER 2	**LITERATURE REVIEW**		**7**
	2.1	Vortex Shedding	7
		2.1.1 Von Karman Vortex Street	10
		2.1.2 Vortex Shedding Frequency	12
	2.2	Reynolds Number	14
	2.3	Flow velocity	16
	2.4	Pressure Loss	17
	2.5	Drag and Lift Force	18

	2.6	Bluff Body	21
	2.7	Different Bluff Body Shapes	22
		2.7.1 Flow around Circular Shape	23
		2.7.2 Flow around Triangular Shape	25
		2.7.3 Flow around Rectangular Shape	26
	2.8	Bluff Body with Splitter	27
	2.9	Computational Fluid Dynamics (CFD)	28
		2.9.1 Turbulence model	30
CHAPTER 3	**METHODOLOGY**		**32**
	3.1	Research Flow Chart	33
	3.2	Model Configuration	34
	3.3	Research Model	36
		3.3.1 ANSYS Design Modeler	36
		3.3.2 Meshing	37
		3.3.3 Boundary Condition	41
		3.3.4 Setup	43
		3.3.5 Solution	44
		3.3.6 Result / Post Processing	44
CHAPTER 4	**RESULTS AND DISCUSSION**		**45**
	4.1	Introduction	45
	4.2	Analysis on Flow around Circular Cylinder	45
		4.2.1 Effect on Pressure Loss	46
		4.2.2 Effect on Drag Force	47
		4.2.3 Effect on Flow Velocity	48
		4.2.4 Effect on Lift Force	51
	4.3	Analysis on Flow around Rectangular	52
		4.3.1 Effect on Pressure Loss	52
		4.3.2 Effect on Drag Force	53
		4.3.3 Effect on Flow Velocity	54
		4.3.4 Effect on Lift Force	57
	4.4	Analysis on Flow around Equilateral Triangular	57
		4.4.1 Effect on Pressure Loss	58

	4.4.2	Effect on Drag Force	58
	4.4.3	Effect on Flow Velocity	60
	4.4.4	Effect on Lift Force	61
	4.5	Analysis on Flow around Bluff Body with Splitter	62
	4.6	Comparison among three bluff body shapes	65
CHAPTER 5	**CONCLUSION AND RECOMMENDATION**		**68**
	5.1	Conclusion	68
	5.2	Recommendation	69
	REFERENCES		71
	APPENDIX		75

LIST OF FIGURES

2.1	Vortex created by the passage of aircraft wing, which is exposed by the colored smoke	8
2.2	Vortex shedding evolving into a vortex street.	8
2.3	Vortex-formation model showing entrainment flows	9
2.4	Von Karman Vortex Street at increasing Reynolds Numbers	10
2.5	Von Karman Vortex Street, the pattern of the wake behind a cylinder oscillating in the Re = 140	11
2.6	Karman Vortex Street phenomenon	12
2.7	Effect of Reynolds number on wake of circular cylinder	16
2.8	Pressure coefficient distributions on cylinder surface compared to theoretical result assuming ideal flow	18
2.9	Oscillating drag and lift forces traces	19
2.10	Schematic of key features in a low dilatation ratio bluff-body flow; time averaged velocity profiles	22
2.11	Streamlines around a rectangular cylinder at moderate Reynolds numbers – based on experimental flow visualizations	27
3.1	Project flow chart	33
3.2	CFD modeling overview	34
3.3	Geometry of three types of bluff body shapes	35
3.4	Computational geometry and boundary condition	36
3.5	Geometry Process – Design Modeler	36
3.6	Meshing of circular bluff body	37
3.7	An unstructured tetrahedral mesh around a circular cylinder	38
3.8	Meshing of rectangular	38
3.9	Meshing of equilateral triangular	39
3.10	Meshing of circular with splitter	39

3.11	Meshing of rectangular with splitter	40
3.12	Meshing of equilateral triangular with splitter	40
3.13	Label of boundary type for inlet	41
3.14	Label of boundary type for outlet	42
3.15	Label of boundary type for circular	42
3.16	Label of boundary type wall	42
4.1	Pressure coefficient distribution around circular (Re = 10 000)	46
4.2	Pressure coefficient with different Reynolds number for rectangular	46
4.3	Drag force with different Reynolds number for circular	47
4.4	Drag coefficient with different Reynolds number for circular	48
4.5	Vector plot colored by the streamwise velocity circular	49
4.6	Contours of velocity magnitude at Re = 5000	50
4.7	Contours of velocity magnitude at Re = 10000	50
4.8	Contours of velocity magnitude at Re = 15000	50
4.9	Contours of velocity magnitude at Re = 20000	50
4.10	Lift force with different Reynolds number for circular	51
4.11	The drag and lift coefficient of a stationary circular	52
4.12	Pressure coefficient with different Reynolds number for rectangular	53
4.13	Drag force with different Reynolds number for rectangular	54
4.14	Drag coefficient with different Reynolds number for rectangular	54
4.15	Vector plot colored by the streamwise velocity for rectangular	55
4.16	Contours of velocity magnitude at Re = 5000	56
4.17	Contours of velocity magnitude at Re = 10 000	56
4.18	Contours of velocity magnitude at Re = 15 000	56
4.19	Contours of velocity magnitude at Re = 20 000	56
4.20	Lift force with different Reynolds number for rectangular	57
4.21	Pressure coefficient with different Reynolds number for equilateral triangular	58
4.22	Drag force with different Reynolds number for equilateral triangular	59
4.23	Drag coefficient with different Reynolds number for triangular	59

4.24	Vector plot colored by the streamwise velocity of equilateral triangular	60
4.25	Contours of velocity magnitude at Re = 5000	60
4.26	Contours of velocity magnitude at Re = 10000	61
4.27	Contours of velocity magnitude at Re = 15000	61
4.28	Contours of velocity magnitude at Re = 20000	61
4.29	Lift force with different Reynolds number for triangular	62
4.30	Contours of velocity magnitude for circular with splitter	63
4.31	Contours of velocity magnitude for rectangular with splitter	63
4.32	Contours of velocity magnitude for equilateral triangular with splitter	64
4.33	Drag coefficient with and without splitter of each bluff body	64
4.34	Pressure loss coefficient with and without splitter of each bluff body	65

LIST OF TABLES

2.1	Classification of disturbance free flow regime	15
2.2	Drag coefficients Cd of various two-dimensional bodies for Re >10^4	20
2.3	Flow regime around smooth, circular cylinder in steady current	24
2.4	Characteristics of Laminar and turbulent flow in pipes	28
3.1	Dimension on different bluff bodies	35
3.2	Number of element and node for circular cylinder	38
3.3	Number of element and node for rectangular	38
3.4	Number of element and node for equilateral triangular	39
3.5	Number of element and node for circular with splitter	39
3.6	Number of element and node for rectangular with splitter	40
3.7	Number of element and node for equilateral triangular with splitter	40
3.8	Boundary condition	41
3.9	Solutions method details for Fluent	43
3.10	Drag coefficient of flow around circular cylinder	44

LIST OF SYMBOL

St	Strouhal Number
f	Frequency
d	Width of shedding body
U	Fluid velocity
Re	Reynolds Number
$V\rho$	Inertial force
μ	Free stream velocity
D	Diameter of the bluff-body
v	Kinematic viscosity of the fluid
C_p	Pressure coefficient
C_d	Drag coefficient
F	Amount of pressure
A	Area

LIST OF APPENDICES

A	Analysis Data on Circular	75
B	Analysis Data on Rectangular	80
C	Analysis Data on Equilateral Triangular	85
D	Project Gantt Chart	90

CHAPTER 1

INTRODUCTION

1.1 Research Background

Understanding the phenomenon vortex shedding in various shapes is utmost important because of the physical applications and it might cause severe damage. For example, the Tacoma Narrows Bridge was found collapse due to the occurrence of vortex shedding. The combination of the aerodynamics of bridge deck cross-section and the wind has been known to produce significant oscillations (Taylor, 2011). The collapse of the bridge over the Tacoma Narrows in 1940 was caused by aero-elastic which involves periodic vortex shedding. The bridge started torsional oscillations, which developed amplitudes up to 40° before it broke. In tall building engineering, vortex shedding phenomenon is carefully taken into account. For example, the Burj Dubai Tower, the world’s highest building, is purposely shaped to reduce the vortex-induced forces on the building (Ausoni, 2009; Baker et al., 2008).

Vortex shedding can be defined as periodic detachment pairs of alternating vortices that bluff-body immersed in a fluid flow, generating oscillating flow that occurs when fluid such as air or water flow past cylinder body at a certain velocity, depending on the size and shape of the body. At Reynolds numbers above 10 000 the flow around a circular cylinder can be separated into at least four different regimes such as subcritical, critical, supercritical and transcritical.

Vortex shedding is one of the most challenging phenomenons in turbulent flows. Hence, the frequency of shedding vortices in the wakes of a bluff body is used to measure the flow rate in vortex parameter. The shape of the bluff body in a stream determines how efficiently it will form vortices. The impact of this study was to optimize the performance of flow meters, such as flow velocity, pressure loss, drag force and lift force in time response by designing a bluff body which generates vortices over a wide range of Reynolds numbers. By completing a vortex shedding analysis, engineers can evaluate whether more efficient structures can and should be developed.

1.2 Problem Statement

A deep understanding of a phenomenon determines the successful of a design. Like any phenomenon such as vortex shedding, more research need to be done carefully to avoid damage or failure is inevitable. Although vortex meter has been known and many research done over the years, the nature of this vortex meter is still not fully recognized yet. It should be noted that many factors affect the phenomenon is defined worse (Pankanin, 2007).

Based on principle working of vortex flow meter, the speed of incoming flow affect to the measurement of the vortex shedding frequency and an unsteady phenomenon flow over a bluff body. Therefore, the bluff body shape plays an important role in determining the performance of a flow meter. Predicated on research conducted by Gandhi et al. (2004), the size and shape of bluff body strongly influence the performance of the flow meter and Lavish Ordia et al. (2013) also supported the statement by adding the least dependence Strouhal number on the Reynold number and minimum power will provide the stability of vortex.

According to Błazik-Borowa et al. (2011), although many methods have been proposed over the years to control dynamics of wake vortex, unfortunately the turbulence models still do not properly described the turbulent vortex shedding phenomenon. The calculations results are not properly for all cases. Sometimes errors occur in a small part of calculation domain only, but sometimes the calculation results are completely incorrect. Thus the computer calculation shall be checked by comparison with measurements results which should include the components of velocity and their fluctuations apart from averaged pressure distribution.

The researches about geometrical shape of bluff body have been conducted by many researchers. However, the flow around circular, rectangular and triangular have been choose to get better understand fluid dynamics and related accuracy of numerical modeling strategies. The research was carried out with various bluff body shapes to identify an appropriate shape which can be used for optimize the configuration of the bluff body on the performance of flow meter

1.3 Objective of Study

The objectives of this research are:

i. To investigate the effect of bluff body shape on the pressure loss, drag force, flow velocity and lift force in the time response (unsteady state).
ii. To investigate the effect of bluff body attached with splitter plate on the flow characteristics.

1.4 Research Scope

The scopes can be summarized as:

i. Develop the Computational Fluid Dynamics (CFD) model for the flow simulation of a bluff body.
ii. Using three different shapes of bluff body which are circular, rectangular and equilateral triangular
iii. Cross-sectional area of the bluff bodies $0.0079m^2$ for circular, $0.015m^2$ for rectangular and $0.0087m^2$ for equilateral triangular.
iv. Dimension of the splitter plate is 0.075m
v. Test at different Reynolds Number ranging from 5000 to 20000.

1.5 Significant of Study

Requirement to optimize the configuration of the bluff body is very important in the flow meter performance, especially in terms of size and shape of the bluff body. To produce the size and design of an appropriate bluff body in order to get the optimum configuration bluff body, deeper study should be carried out from various aspects.

CHAPTER 2

LITERATURE REVIEW

This chapter introduces the fundamentals of the main topics forming the basis of the current study. In order to fully understand the vortex shedding in various shapes, there must be a deeper understanding of the forces involved. Furthermore the vortex shedding phenomenon may affect the bluff body shapes. This literature will introduce vortex shedding, Von Karman Vortex Street, vortex shedding frequency, Reynolds number, flow velocity, pressure loss, drag and lift force and bluff body.

2.1 Vortex Shedding

One of the first to describe the vortex shedding phenomenon was Leonardo da Vinci by drawn some rather accurate sketches of the vortex formation in the flow behind bluff bodies. The formation of vortices in body wakes is described by Theodore von Karman (Ausoni, 2009). In fluid dynamics, vortex is district, in a fluid medium, where the flow, most of which revolve on the vortical flow axis, occurring either direct-axis or curved axis. In other words, vortex shedding occurs when the current flow of a water body is impaired by an obstruction, in this case a bluff body. Vortex or vortices are rotating or swirl, often turbulent fluid flow. Examples of a vortex or vortices appear in Figure 2.1. Speed is the largest at the center, and reduces gradually with distance from the center

Figure 2.1: Vortex created by the passage of aircraft wing, which is exposed by the colored smoke (Rafiuddin, 2008)

Vortex shedding has become a fundamental issue in fluid mechanics since shedding frequency Strouhal's measurement in 1878 and analysis of stability of the Von Karman vortex Street in 1911 (Chen & Shao, 2013). The phenomenon of vortex formation and shedding has been deeply study by many researchers included (Gandhi et al., 2004; Ausoni, 2009; Azman, 2008). In the natural vortex shedding and vortex-street appears established when stream flow cross a bluff body can be seen in Figure 2.2. It has been observed that vortex shedding is unsteady flow which creates a separate stream throughout most of the surface and generally have three types of flow instability namely, boundary layer instability and separated shear layer instability (Gandhi et al., 2004).

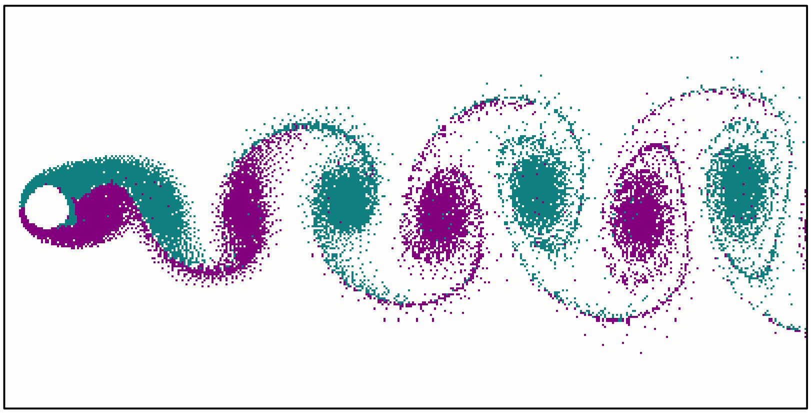

Figure 2.2: Vortex shedding evolving into a vortex street.

This region arises when a flow is unable to follow the aft part of an object. As the aft part has certain bluntness the flow detaches initially from the object's surface and a region of back-flow is generated, the so-called separation bubble. Vortex shedding also produced shear layer of the boundary layer. At some point, an increasingly attractive vortex draws the opposing shear layer across the near wake. Signed vorticity opposite approach, cutting off circulation to the vortex again rising, which then drops move downstream.

Vortex shedding is determined by two properties; the viscosity of the fluid passing over a bluff body, and the Reynolds number of that fluid flow. The Reynolds number relates inertial forces to the viscous forces of fluid. In higher Reynolds number regions, inertial forces dominate the flow and turbulence is found, while in low Reynolds number regions, viscous forces dominate the flow and laminar flow is developed. The creation of strong clean vortices occurs in lower Reynolds number regions. (Bjswe et al., 2010). Based on Figure 2.3, fluid a enter the vortex weakens is due to its opposing sign, fluid b enters the feeding shear layer and cuts off the further circulation to the growing vortex while fluid moves back toward the cylinder and it will grow until it is strong enough to draw the upper shear layer across the wake. This process repeats itself and is known as vortex shedding which evolves into a vortex street.

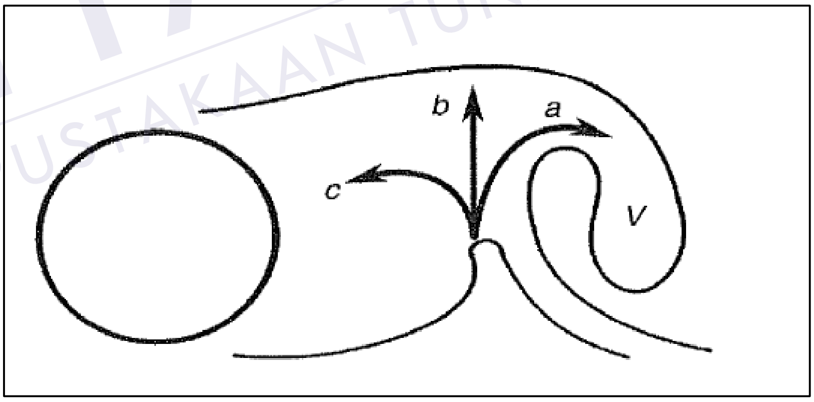

Figure 2.3: Vortex-formation model showing entrainment flows (Ausoni, 2009)

Generally, ordinary study on vortex shedding bluff body are Von Karman Vortex Street, Vortex Induced Vibrations, excitation vortex and vortex shedding characteristics.

2.1.1 Von Karman Vortex Street

Von Karman Vortex is a term defining the detachment periodic alternating pairs of vortices that bluff-body immersed in a fluid flow, generating up swinging, or Vortex Street, behind it, and causing fluctuating forces to be experienced by the object. When a fluid flows over a blunt, 2 dimensional bodies, vortices are created and shed in an alternating fashion on the top and bottom of the body (Graebel, 2007; Bjswe et al., 2010). This phenomenon was initially symmetrical but then it turned into a classical alternating pattern because the body is symmetrical. Figure 2.4 is a good depiction of a common von Karman vortex street. This behavior was denominated Theodore Von Karman for his studies in the field. The Von Karman vortex street is a typical fluid dynamics example of natural instabilities in transition from laminar to turbulent flow conditions The Von Karman Vortex Street is a typical example of the fluid dynamic instability inherent in the transition from laminar to turbulent flow conditions.

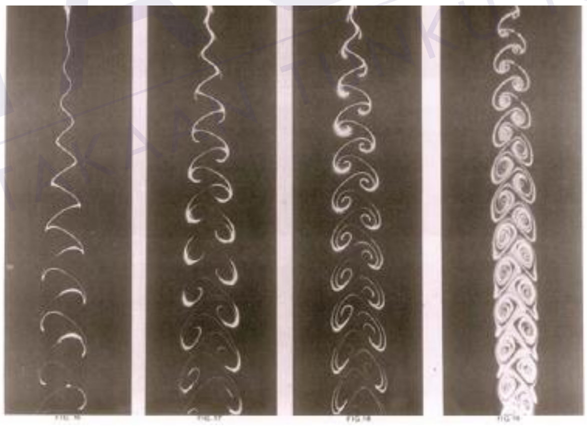

Figure 2.4: Von Karman Vortex Street at increasing Reynolds Numbers (Bjswe et al., 2010)

The first theory of vortex streets in 1911 has been published by Theodore von Karman that examines the model analysis Von Karman Vortex Street for. He found that linear stability has been common for point vortex of finite size and can stabilize the array (Azman, 2008). Even though von Karman's (1912) ideal vortex street has been long associated with the wake of a circular cylinder; the only requirement for the existence of a vortex street is two parallel free shear layers of opposite circulation which are separated by a distance, h. The ideal mathematical description, limited for two-dimensional flows, is predicated on the stability investigation of two parallel vortex sheets with the same but opposite vortices with intensity. Linear vortex intensity that is located at the same distance from each other along the sheets and movement velocity resulting from pushing investigated. The intensity of the vortex is based on circulation and is defined as;

$$\Gamma = \oint \vec{U}.d\vec{r} \tag{2.1}$$

Vortex Street will only be considered at a given range of Reynolds numbers, usually above the limit *Re* of about 90. When the flow reaches Reynolds numbers between 40 to 200 in the wake of the cylinder, alternated vortices are emitted from the edges behind the cylinder and dissipated slowly along the wake as shown in Figure 2.5

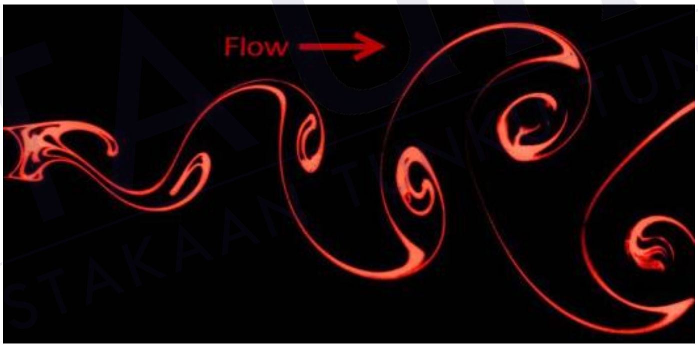

Figure 2.5: Von Karman Vortex Street, the pattern of the wake behind a cylinder oscillating in the Re = 140 (Aref et al., 2006; Azman, 2008)

The vortex flow meter is based on the well-known von Karman vortex street phenomenon. This phenomenon consists on a double row of line vortices in a fluid. Figure 2.6 show the under certain conditions which Karman Vortex Street spilled in the center of the cylinder body lies when the fluid velocity is relatively perpendicular to the cylinder generator. A remarkable phenomenon because the flow direction of the oncoming flow may be perfectly steady when the occurrence of periodic shedding of eddies. Vortex streets can often be visually perceived, for example, each successful meter design is determined by comprehensive understanding of applied physical phenomena. Von Karman vortex street phenomenon is very intricate and sensitive on numerous physical factors.

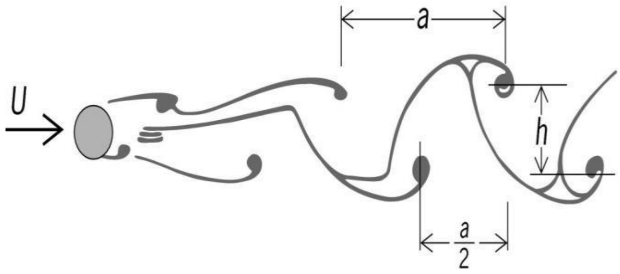

Figure 2.6: Karman Vortex Street phenomenon (Kulkarni et al., 2014)

Vortex flow meter is still very attractive for industrial applications due to its high accuracy, is not sensitive to the medium physical properties and linear frequency-dependent than the flow rate. The frequency of the vortex generated is directly proportional to the flow velocity. It should be noted here that many less obvious factors influence the phenomenon. Therefore, the need to apply various research methods for the characterization of the phenomenon of vortex shedding causes is necessity of investigations with application of miscellaneous methods.

2.1.2 Vortex Shedding Frequency

After experimental Strouhal's introduced to determine the vortex shedding frequency, some appropriate methods that have been proposed in the technical literature. Shedding frequency, f, have been identified from the peak in the power spectrum and lift coefficients used to calculate the Strouhal number (Gonçalves et al., 1999). The frequency of vortex shedding becomes constant at lock-in and the free-stream velocity changes the value of the Strouhal number. Vortex shedding frequency is perpendicular to the direction of flow and has a period equal to the lift. Since the vortices are shed periodically, resulting in lift on the body also vary from time to time.

The Strouhal number (St) is a dimensionless proportionality constant between the main frequency vortex shedding and the free stream velocity divided by bluff-body dimensions. It is also often approximated by a constant value.

REFERENCES

Almeida, O., Mansur, Silveria-Neto. On The Flow Past Rectangular Cylinders : Physical Aspects And Numerical Simulation. *Thermal Engineering*. 2008. v.7, pp.55-64.

Muhammad Tedy Asyikin. *CFD Simulation of Vortex Induced Vibration of a Cylindrical Structure*. Master Thesis. Norwegian University of Science and Technology; 2012.

Ausoni, P. *Turbulent Vortex Shedding from a Blunt Trailing Edge Hydrofoil*. Ph. D. Thesis. Ecole Polytechnique Federale de Lausanne; 2009.

Rafiuddin Azman. *Study of Vortex Shedding Around Bluff Body Using Air Flow Test Rig*. Bachelor Thesis. Universiti Malaysia Pahang; 2008.

Badami, P., Shrivastava, V., Saravanan, V., Hiremath, N., & Seetharamu, K. N. Numerical Analysis of Flow past Circular Cylinder with Triangular and Rectangular Wake Splitter. *World Academy of Science, Engineering and Technology*. 2012. *6*(9), 382–389.

Baker, W. F., Korista, D. S., & Novak, L. C. (2008). Engineering the world's tallest–Burj Dubai. Proceedings of CTBUH 8th world congress "Tall & green: typology for a sustainable urban future", Dubai (pp. 3-5).

Berrone, S., Garbero, V., & Marro, M. (2011). Numerical simulation of low-Reynolds number flows past rectangular cylinders based on adaptive finite element and finite volume methods. *Computers & Fluids*, *40*(1), 92–112. http://doi.org/10.1016/j.compfluid.2010.08.014

Bhuyan, M. S., Othman, M., Ali, S. H. M., Majlis, B. Y., & Islam, M. S. (2012, September). Modeling and simulation of fluid interactions with bluff body for energy harvesting application. *In Semiconductor Electronics (ICSE), 2012 10th IEEE International Conference on (pp. 123-127). IEEE.*

Bjswe, P., Johnson, B., & Phinney, B. *Optimization of Oscillating Body for Vortex Induced Vibrations*. Worcester Polytechnic Institute; 2010.

Błazik-Borowa, E., Bęc, J., Nowicki, T., Lipecki, T., Szulej, J., & Matys, P. (2011). Measurements of flow parameters for 2-D flow around rectangular prisms of square and rectangle cross-sections located on the ground. *Archives of Civil and Mechanical Engineering*, *11*(3), 533–551.

Chan, A. S., Co-adviser, A. J., Maccormack, R., & Gumport, P. J. (2012). Control and suppression of laminar vortex shedding off two-dimensional bluff bodies.

Chen, Y. J., & Shao, C. P. (2013). Suppression of vortex shedding from a rectangular cylinder at low Reynolds numbers. *Journal of Fluids and Structures*, *43*, 15–27. http://doi.org/10.1016/j.jfluidstructs.2013.08.001

Dar, B. (2013). 2D Flow around a Rectangular Cylinder : A Computational Study. *An International Journal of Science and Technology Bahir Dar, Ethiopia*, *2*(1), 1–26.

Du, Y., Qian, R., & Peng, S. (2006). Coherent structure in flow over a slitted bluff body. *Communications in Nonlinear Science and Numerical Simulation*, *11*(3), 391–412. http://doi.org/10.1016/j.cnsns.2004.07.003

Gandhi, B. K., Singh, S. N., Seshadri, V., & Singh, J. (2004). Effect of bluff body shape on vortex flow meter performance. *Indian Journal for Engineering and Material Sciences*, *11*, 378-384.

Herbert, C. G., & Edson, D. R. V. (1999). Strouhal Number Determination For Several Regular Polygon Cylinders For Reynolds Number Up to 600. In *15th Brazilian Congress of Mechanical Engineering* (Vol. 7).

Gonçalves, H. C., & Vieira, E. D. R. (1999). Strouhal Number Determination for Several Regular Polygon Cylinders for Reynolds Number up to 600. In *Proceedings of the COBEM* (Vol. 99).

Igarashi, T. (1999). Flow Resistance and Strouhal Number of a Vortex Shedder in a Circular Pipe. *Transactions of the Japan Society of Mechanical Engineers Series B*, *65*(629), 229–236. http://doi.org/10.1299/kikaib.65.229

Igbalajobi, a., McClean, J. F., Sumner, D., & Bergstrom, D. J. (2013). The effect of a wake-mounted splitter plate on the flow around a surface-mounted finite-height circular cylinder. *Journal of Fluids and Structures*, *37*, 185–200. http://doi.org/10.1016/j.jfluidstructs.2012.10.001

Joubert, E. C., Harms, T. M., & Venter, G. (2015). Computational simulation of the turbulent flow around a surface mounted rectangular prism. *Journal of Wind Engineering and Industrial Aerodynamics*, *142*, 173–187. http://doi.org/10.1016/j.jweia.2015.03.019

Kulkarni, A. A., Harne, M. S., & Bachal, A. (2014). Study of Vortex Shedding Behind Trapezoidal Bluff Body by Flow Visualization Method. In *International Journal of Engineering Research and Technology* (Vol. 3, No. 9 (September-2014)). ESRSA Publications.

Laroussi, M., Djebbi, M., & Moussa, M. (2014). Triggering vortex shedding for flow past circular cylinder by acting on initial conditions: A numerical study. *Computers & Fluids*, *101*, 194–207. http://doi.org/10.1016/j.compfluid.2014.05.034

Liaw, K. F. *Simulation of Flow around Bluff Bodies and Bridge Deck Sections using CFD*. Ph. D Thesis. University of Nottingham; 2005.

Liu, Z., & Kopp, G. a. (2012). A numerical study of geometric effects on vortex shedding from elongated bluff bodies. *Journal of Wind Engineering and Industrial Aerodynamics*, *101*, 1–11. http://doi.org/10.1016/j.jweia.2011.11.007

Mastenbroek, J. (2010). B Bluff body flow: wake behavior behind a heated circular cylindeer.

Lavish Ordia, A. V., Agrawal, A., & Prabhu, S. V. (2013). Influence of After Body Shape on the Performance of Blunt Shaped Bodies as Vortex Shedders, *7*(10), 836–841.

Ou, Z. (2007). *Numerical Simulation of Flow around Vertical Cylinders*. University of Western Australia.

Pankanin, G. L. (2007). Experimental and Theoretical Investigations Concerning the Influence of Stagnation Region on Karman Vortex Shedding. In *2007 IEEE Instrumentation & Measurement Technology Conference IMTC 2007* (pp. 1–6). IEEE. http://doi.org/10.1109/IMTC.2007.379231

Peng, J., Fu, X., & Chen, Y. (2008). Experimental investigations of Strouhal number for flows past dual triangulate bluff bodies. *Flow Measurement and Instrumentation*, *19*(6), 350–357. http://doi.org/10.1016/j.flowmeasinst.2008.05.002

Prajapati, C. B., Delhi, N., Singh, S. N., Patel, V. K., & Seshadri, V. (2010). CFD analysis of permanent pressure loss for different types of flow meters in industrial applications. In *37th National and 4th International conference of fluid mechanics & fluid power* (pp. 1-8)

Prasad, A. and Williamson, C. H. K. (1997). The Instability of the Shear Layer Separating from a Bluff Body. *Journal of Fluid Mechanics*, *333*, 375–402.

Pun, C. W., & Neill, P. L. O. (2007). Unsteady flow around a Rectangular Cylinder. In *16th Australasian Fluid Mechanics Conference Crown Plaza* (pp. 1449–1456).

Schewe, G. (2013). Reynolds-number-effects in flow around a rectangular cylinder with aspect ratio 1:5. *Journal of Fluids and Structures*, *39*, 15–26. http://doi.org/10.1016/j.jfluidstructs.2013.02.013

Shanbhogue, S. J. (2008). *Dynamics Of Perturbed Exothermic Bluff-Body Dynamics Of Perturbed Exothermic Bluff-Body Flow-Fields*. Georgia Institute of Technology.

Spiteri, M. *Aerodynamic Control of Bluff Body Noise*. Ph. D Thesis. University of Southampton; 2011.

Steggel, N. *A Numerical Investigation of the Flow Around Rectangular Cylinders*. Ph. D Thesis. University of Surrey; 1998.

Sudhakar, Y., & Vengadesan, S. (2012). Vortex shedding characteristics of a circular cylinder with an oscillating wake splitter plate. *Computers and Fluids*, *53*(1), 40–52. http://doi.org/10.1016/j.compfluid.2011.09.003

Taylor, Z. J. *Vortex shedding from elongated bluff bodies*. Ph. D Thesis. University of Western Ontario; 2011.

Taylor, Z. J., Gurka, R., & Kopp, G. a. (2014). Effects of leading edge geometry on the vortex shedding frequency of an elongated bluff body at high Reynolds numbers. *Journal of Wind Engineering and Industrial Aerodynamics*, *128*, 66–75. http://doi.org/10.1016/j.jweia.2014.03.007

Yunus A, C., & John M, C. (2006). *Fluid Mechanic :Fundamental and Applications* (2nd ed.). New York: McGraw-Hill.

Zhang, X., & Perot, B. (2000). Turbulent Vortex Shedding From Triangle Cylinder Using the Turbulent Body Force Potential Model. In *Proceedings of ASME Fluids Engineering Division, FEDSM2000-11172* (pp. 1–6).

Zheng, Z. C., & Zhang, N. (2008). Frequency effects on lift and drag for flow past an oscillating cylinder. *Journal of Fluids and Structures*, *24*(3), 382–399. http://doi.org/10.1016/j.jfluidstructs.2007.08.010