11th International Conference on Marine Technology
MARTEC 2018

VORTEX INDUCED VIBRATION ON COLD WATER PIPE OF OTEC

Muhammad Zaki Prawira[1*], Jaswar Koto[1]

[1]School of Mechanical Engineering,
Faculty of Engineering,
Universiti Teknologi Malaysia,
81310 UTM Johor Bahru, Johor Bahru,
Malaysia

ABSTRACT

OTEC is a process that can produce electricity by using the temperature difference between deep cold around 28 degree Celsius ocean water and warm tropical surface waters around 6 degree Celsius. Indonesia have many potential to build OTEC, among them is in Sumatera Utara, Bali, Flores Sea, Makassar Strait. The problem with pipes in the ocean is that as the ocean water flows past the pipe with a certain current velocity, a phenomenon called vortex-induced vibrations (VIV) might occur, potentially leading to very large oscillations. As the OTEC cold water pipe has unusual large diameter, the fluid will flow at unusual high Reynolds numbers. This paper investigates the effect flow past the pipe with different velocities and large diameters of pipe. The location is in Makassar strait and the velocity is obtained by data from previous researcher based on deep water is 0.8 m/s, 0.62 m/s, 0.46 m/s , 0.35 m/s , 0.28 m/s. The large diameter of pipe is obtained used OTEC Pro Simulation based on population in Karampuang Island(Makassar strait) is 4m and 3m diameter of pipe. The result obtained from numerical simulation used ANSYS FLUENT with show the Cl, Cd , Fv and flow pattern.

Keywords : *Flow Around Cylinder, OTEC, Drag Coefficient, Lift Coefficient, ANSYS*

1.0 INTRODUCTION

Indonesia is the tropical oceans country, approximately defined by latitudes less than 20 degrees, may be thought of as enormous passive solar collectors. Indonesia has 77 % of total area covered by the ocean [3]. OTEC can be done effectively and on a large scale to provide a source of renewable energy that is needed to cover a wide range of energy issues. Floating ocean thermal energy conversion (OTEC) system usually consist an up to 500 m long cold water pipe (CWP) and used large diameter. Renewable energy from OTEC is potential for a power source for small islands to larger islands in Indonesia, the potential location for OTEC is in South Kalimantan, East Kalimantan, Makassar Strait, Timur Strait, and Morotai Island [7].

Wind, sea current and ocean surface waves is a concern for in floating ocean thermal energy conversion (OTEC) system design. Important to study the addiction of the cold water pipe (CWP) on the effect of drag amplification due to vortex induced vibrations, relative velocity of the flow around the pipe [2]. Model and simulation test data for cold water pipe (CWP) are necessary for the update. It is also useful if computational fluid dynamics (CFD) analysis is available for the

*Corresponding author: zakiprawira30@gmail.com

resolve of confirmation. Based on my introduction, this study focuses on flow around cylinder with condition environment in Makassar Strait use computational fluid dynamics (CFD).

2.0 METHODOLOGY

Research methodology is essential to keep the research on the track and ensure the consistency of the result with the objective need to achieve. The first step is collect data from literature review to get parameter of cold water pipe and parameter of environmental. For modeling cold water pipe will performed with Design Modeler Geometry and for performance analysis will performed with ANSYS fluent with turbulent model used K-ε Realizable and Pressure-Velocity Coupling Scheme used PISO. Figure 1 show the step of methodology.

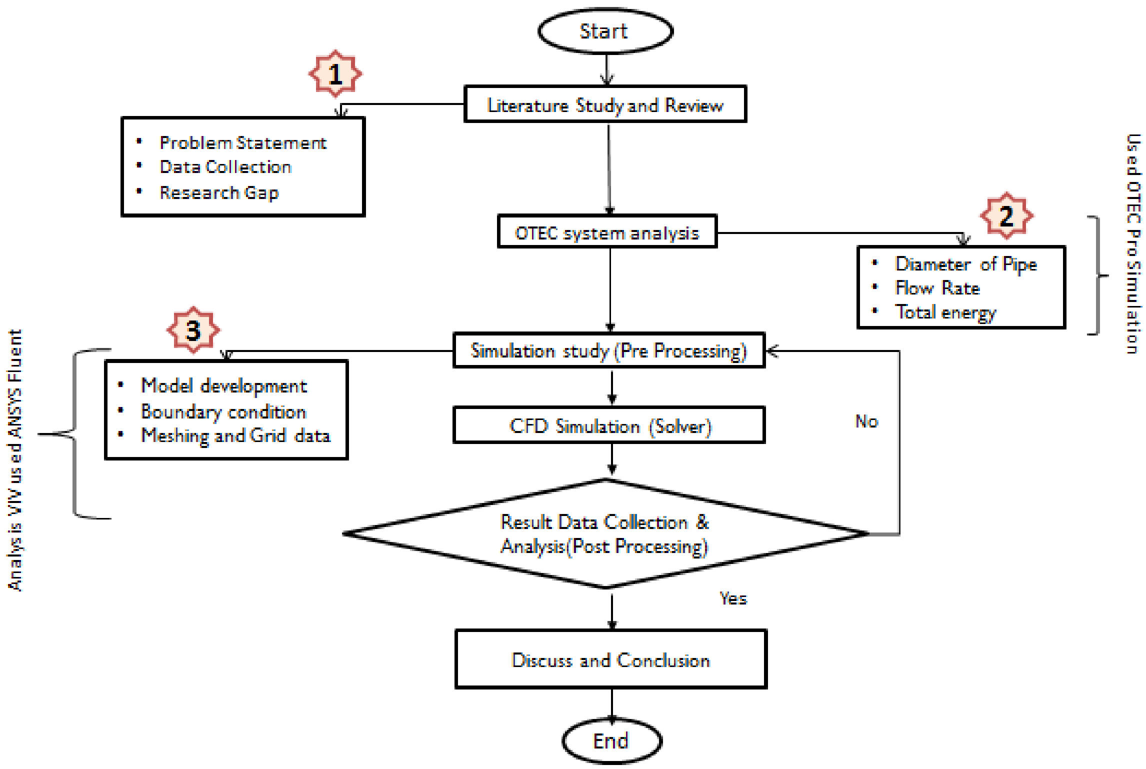


Figure 1: Flow Diagram of Methodology

2.2 Drag and Lift Force

Friction can also contribute to this force. These forces can be divided into two types which are lift for and drag force. As mentioned in the previous section, shedding vortices will cause irregular and periodic changed pressure distribution that contributes to the external and resultant force on the body of cylindrical structure [6]. Lift force moves in cross-flow direction while drag force moves in in-line direction as shown in figure 2.

Drag force, FD and Lift force, FL are formulated:

$$F_D = C_D \frac{1}{2} \rho D L U^2 \tag{1}$$

$$F_L = C_L \frac{1}{2} \rho D L U^2 \tag{2}$$

C_D is Drag Coefficient ,C_L is Lift Coefficient ,ρ is Fluid density ,L is Length of cylinder ,D is Cylinder diameter, U is Flow velocity,

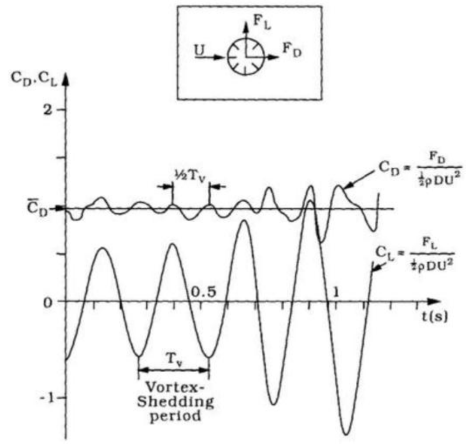


Figure 2: Oscillating drag and lift forces traces, adapted from Sumer

2.2 PARAMETER AND ENVIRONMENTAL DATA

The data for sea current profile according to [1] and [5] with assume the data graphic. larger velocity in water depth 100 m is 0.8 m/s for JAS condition, in water depth 200 m is 0.62 m/s for AMJ & JAS condition, in water depth 300 m is 0.46 m/s for JFM condition, in water depth 400 is 0.35 m/s for JFM, in water depth 500 m is 0.28 m/s for JFM condition. Karampuang Island is an island located in Karampuang Village, Makassar Strait, West Sulawesi Province, The area of Karampuang Island is 6.37 km2 with a population around 5000 people and system of OTEC used closed cycle

The diameter of the pipe calculated based on total energy required of OTEC and cold water flow rate. In this study is calculated by software OTEC Pro Simulation. Table 1, 2, 3 shows the parameter data.

Table 1: Model Specifications

Properties	**Dimensions**
Diameter Total in flow rate 100 m/s	3 m
Diameter Total in flow rate 50 m/s	4 m
Inside Diameter of pipe for cold water in flow rate 100 m/s	2.65 m
Inside Diameter of pipe for cold water in flow rate 50 m/s	3.75 m
Wall Thickness in flow rate 100 m/s	0.3 m
Wall Thickness in flow rate 50 m/s	0.2 m

Table 2: Re for Diameter of pipe in 3m

Sea Current Velocity, U (m/s)	**Reynolds Number, Re**	**Temperature (°c)**	**viscosity**	**Sea Current density (kg/m³)**	**Water Depth (m)**

0.8	2.37E+05	21.5	1.04E-03	1024	100
0.62	1.61E+06	16	1.18E-03	1025	200
0.46	1.05E+06	11	1.35E-03	1026	300
0.35	7.29E+05	8	1.48E-03	1027	400
0.28	5.49E+05	6	1.57E-03	1027	500

Table 3: Re for Diameter of pipe in 4m

Sea Current Velocity, U (m/s)	Reynolds Number, Re	Temperature (ºc)	viscosity	Sea Current density (kg/m³)	Water Depth (m)
0.8	3.16E+06	21.5	1.04E-03	1024	100
0.62	2.15E+06	16	1.18E-03	1025	200
0.46	1.40E+06	11	1.35E-03	1026	300
0.35	9.71E+05	8	1.48E-03	1027	400
0.28	7.33E+05	6	1.57E-03	1027	500

2.3 SOLUTION DOMAIN

The solution domain is the abstract environment where the solution is calculated. The shape of the solution domain can be circular or rectangular depending on the characteristics of the solution. Generally, many simulations use a rectangular box shape as the solution domain. The selected solution domain is L = 40D times D = 17D and the center of the cylinder is 10D at x-axis and 8.5D at y-axis. In the selection of solution domain size and shape, it is compulsory to consider the optimum size. If the size of the domain is too small, less time is needed for the solution to be solved in compare to the large scale of solution domain.

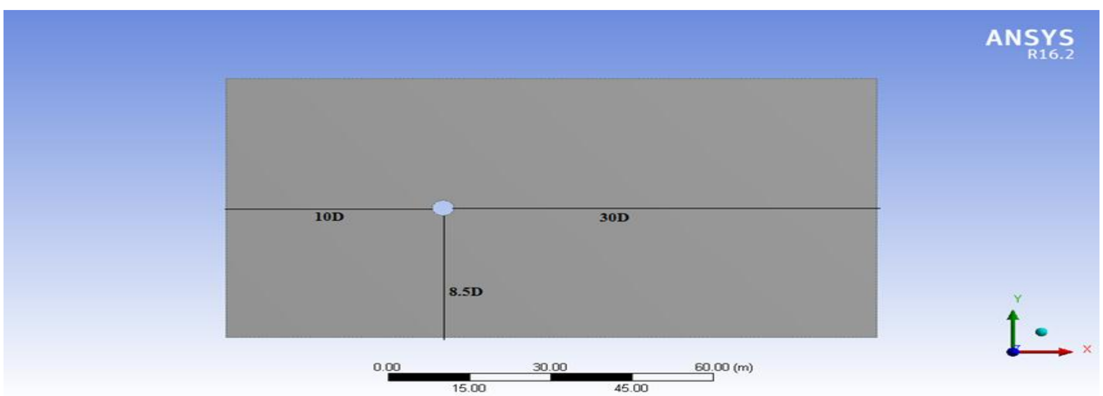


Figure 3: Solution Domain in Design Modeler

2.4 MESHING CONFIGURATION

The next step is meshing configuration. The meshing configuration is clear after solution domain is created. The term meshing and grid generation is interchangeable and has similar literal meaning.

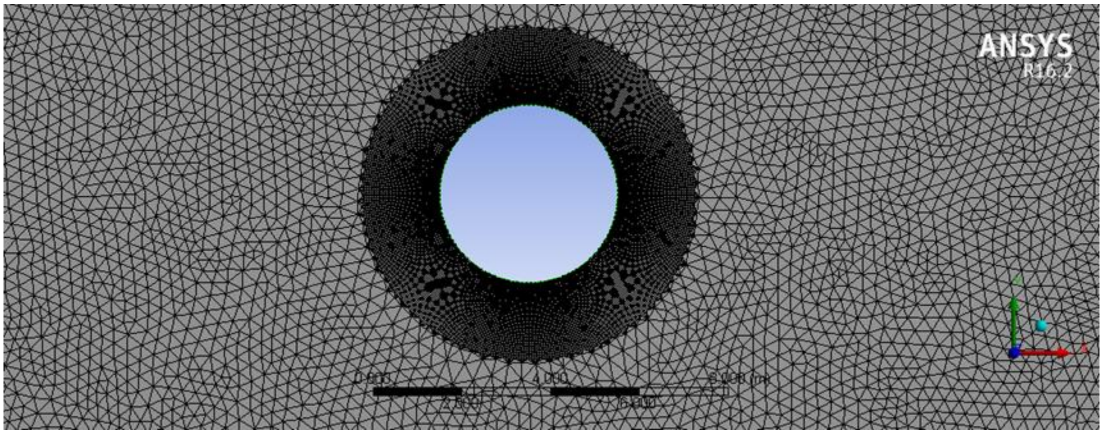


Figure 4: Meshing configuration used ANSYS Fluent.

Table below show statistic meshing in ANSYS Fluent based on current study.

Table 4: Details of meshing configuration

Statistics	Description
Number of Nodes	120348
Number of Elements	213840
Solution Method	Triangular Method

2.5 BOUNDARY CONDITION

Boundary condition, as mentioned previously is the process where variable flow information is specified. Boundary conditions can be classified as inlet, outlet, wall, symmetry, periodic and axis boundaries. Inlet boundary is the condition that allows the flow into the solution domain. The parameters include pressure, velocity and mass flow inlets. The outlet is the condition that allows the flow out of the solution domain.

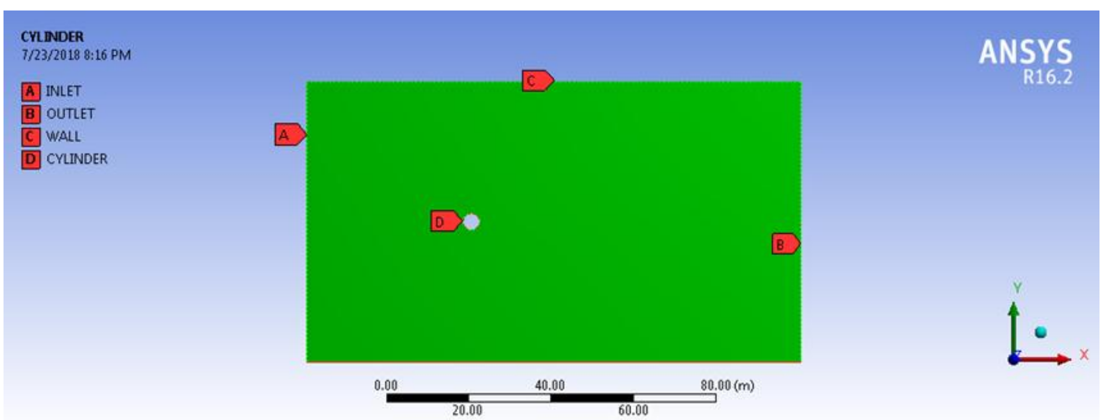


Figure 5: Boundary condition defined in current studies.

3.0 VALIDATION

This section will described the validation of results obtained from simulation previously. In current studies, the method of validation is by comparing the results with Strouhal Number (St).

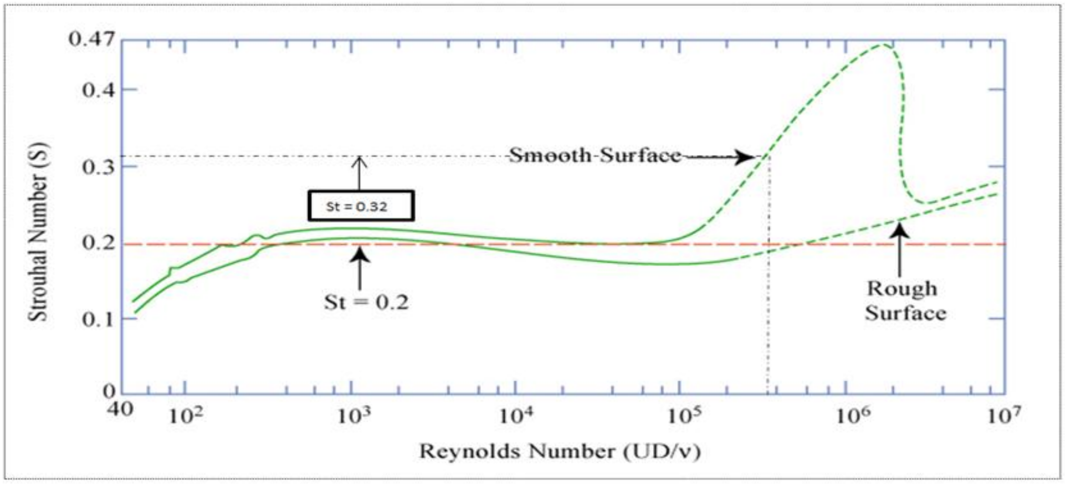


Figure 6: Variation in vortex pattern based on Reynolds number (Re) [4].

From Figure 6, Lienhard specifies the relationship between Strouhal number (St) and Reynolds number (Re). The Re is from the range Re = 0 until 107. In current studies, used Re 100000 until 300000. As per shown in Figure 6, the Strouhal number (St) should be around 0.32 for (Re) 549477.707 at smooth surface. The Fast Fourier Transform (FFT) is implemented to calculate (St).

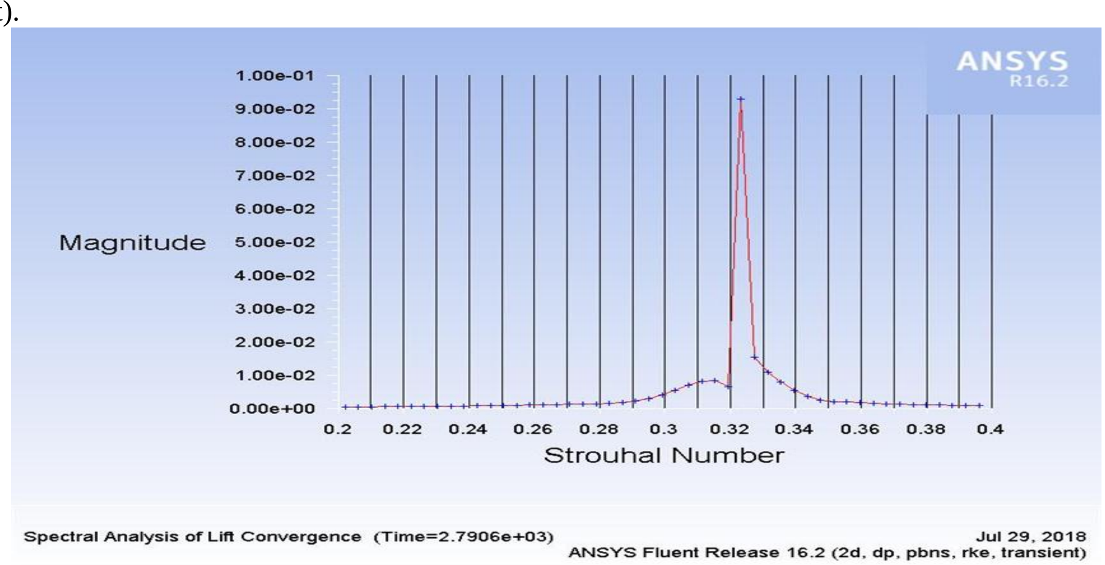


Figure 7: FFT Plot at Re = 549477.707

4.0 RESULTS Cd , Cl , and Fv

From figure 8 the higher Cd for diameter 4m at 500 m is 1.24 and lower is at 100m Cd is 0.904 and the higher Cd for diameter 3m at 500 m is 0.825 and lower is at 100m Cd is 0.69. Because at 500 m the force is bigger so can increase friction Drag and also The Area is mean Diameter can increase for in-line direction.

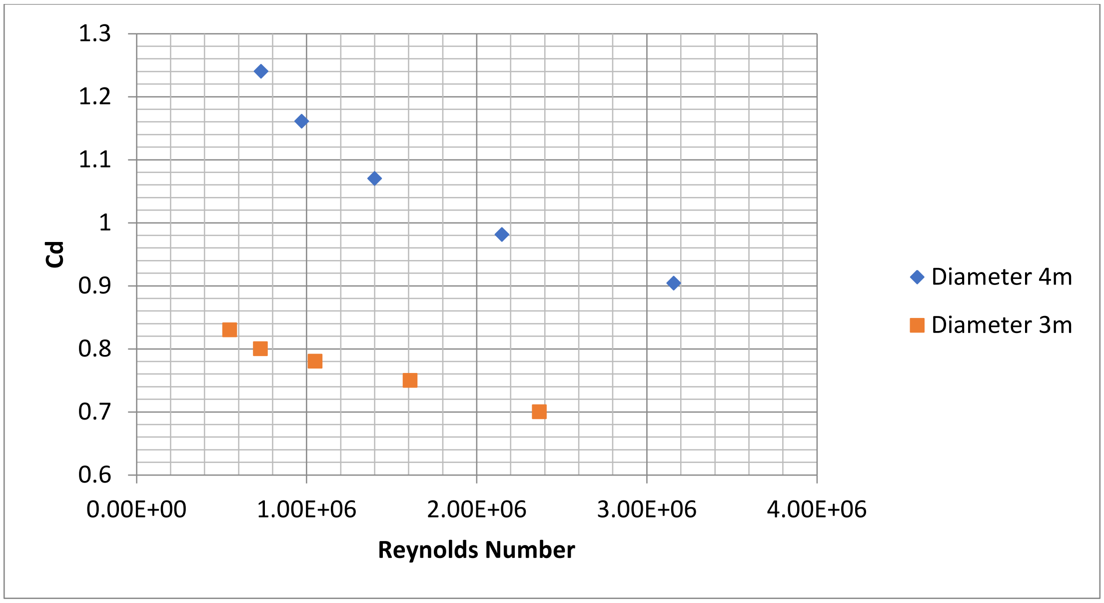


Figure 8: Comparison of Cd

From Figure 9 the higher Cl for diameter 4m at Re 3.16E+06 in 100 m is 0.181 and lower is at 100 m Cl is 0.04 .The higher Cl for diameter 3m at Re 2.37E+05 in 100 m is 0.121 and lower is at 500m Cl is 0.03.

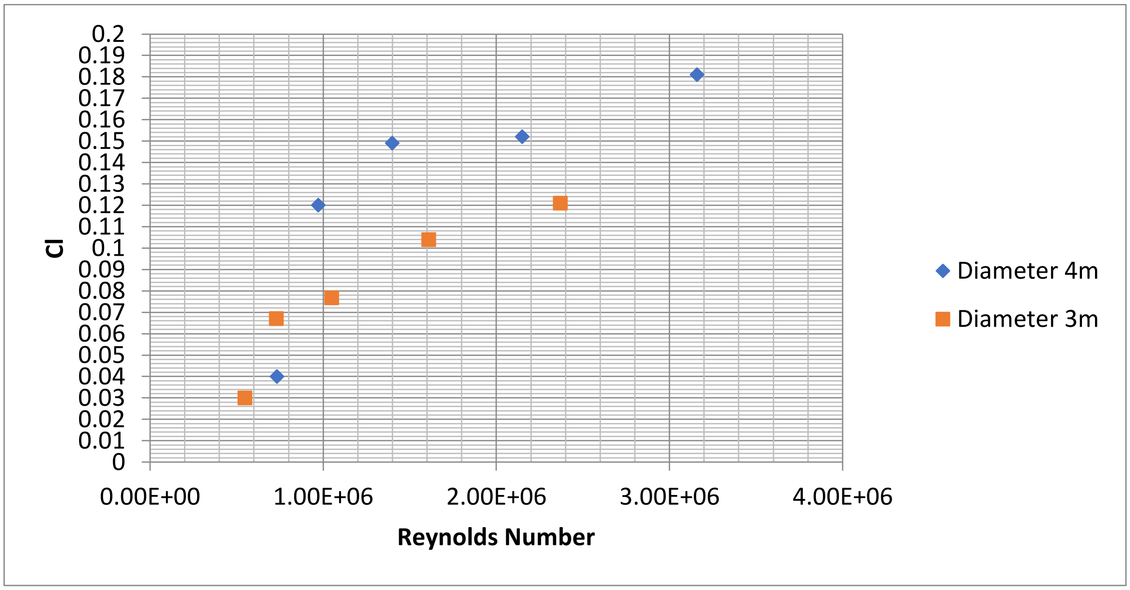


Figure 9: Comparison of Cl

The shedding frequency is obtain from Cl history and The Fast Fourier Transform (FFT) is implemented to calculate the frequencies spectrum of fv. From figure 5.22 is obtained the high Shedding Frequency Fv is at at Re 3.16E+06 in 100 m for diameter 4m is 0.11 Hz . the high Shedding Frequency Fv is at at Re 3.16E+06 in 100 m for diameter 3m is 0.072 Hz.

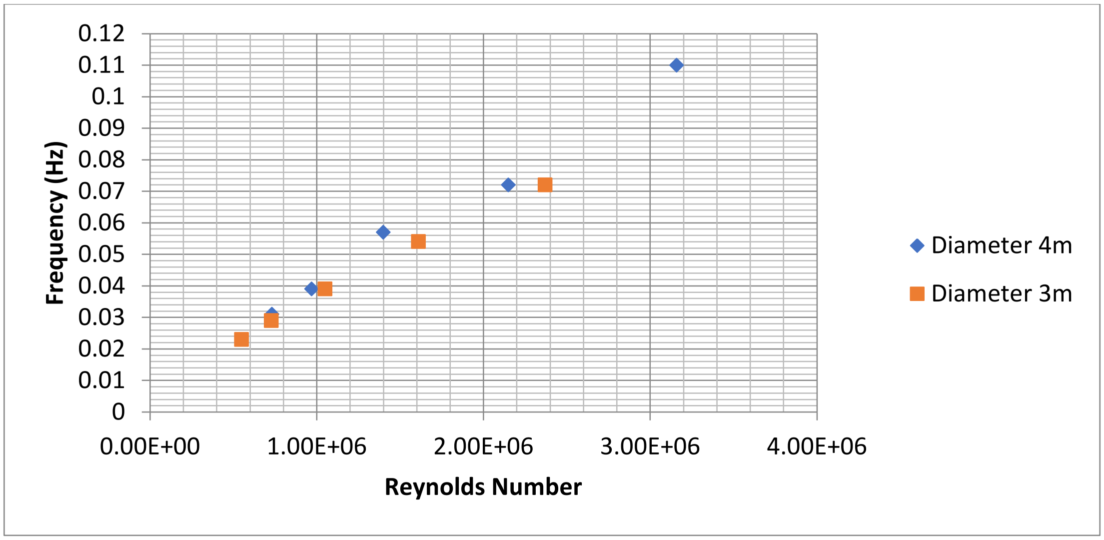
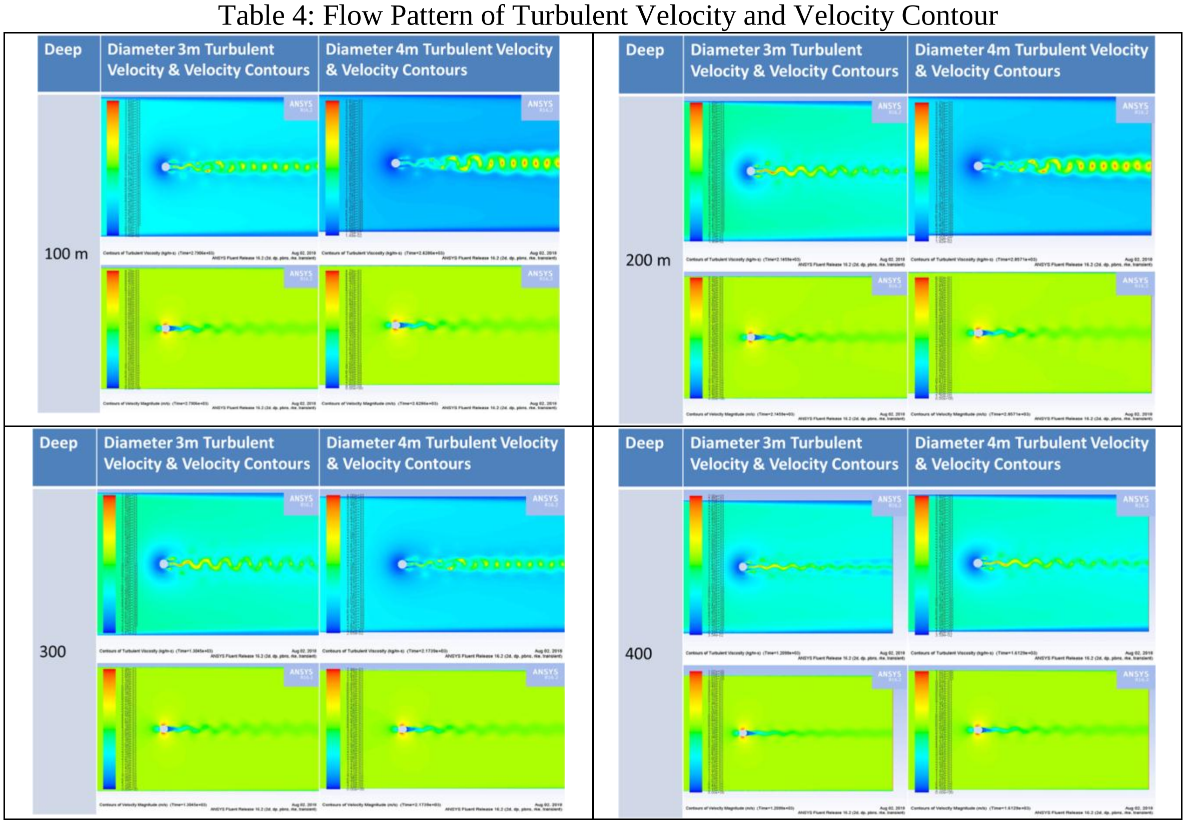


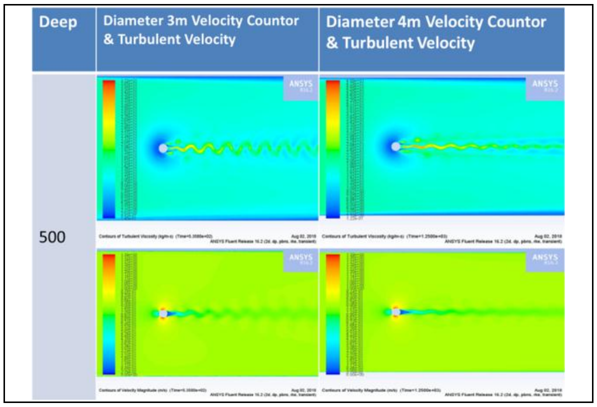
Figure 10: Comparison of Fv

5.1 Flow Pattern of Turbulent Velocity and Velocity Contour

The Turbulent Velocity and Velocity Contour is compared to see the relationship of flow between Diameter 3m and 4m and also velocity at 0.8 m/s in deep 100 m, 0.62 m/s in deep 200 m, 0.46 m/s in deep 300 m, 0.35 m/s in deep 400 m, 0.28 m/s in deep 500 m. The Turbulent Velocity and Velocity Contour as shown in Table 5 is taken at global time lapse to obtain the result where the changes in the contour at unsteady state. We can see a higher turbulence at 0.8 m / s at a depth of 100m and the lower the velocity, the turbulent becomes smaller.

Table 4: Flow Pattern of Turbulent Velocity and Velocity Contour





5.0 CONCLUSION AND RECOMMENDATION

The main purpose of this research study is to determine suitable diameter of cold water pipes for OTEC system in Makassar Strait, Indonesia and to analyze vortex induced vibration of the selected cold water pipes using different Velocity & diameters.

The simulation is done to study the analyze Flow past the cylinder with different velocity and diameter. By Using ANSYS FLUENT, the Reynolds number was performed on cylinder.

Based on the result obtained from the simulation, The suitable diameter for OTEC in Karampuang Island, Makassar strait is 4m and 3m based on flow rate and energy requirement 0.005 MW for 5000 people with the Closed Cycle system. The In-line decreases with the increase of Re otherwise Cross- flow is increase with the increase Re and also increase the shedding frequency. The high vibration is in diameter 4m at Re 3.16E+06 in deep 100m is 0.11 Hz

From the simulation and results outcome, there is an improvement that can be practiced in future work for more accurate and better findings in the analysis:

I. To conduct simulation 3D on the effect of other parameters such as diameter and Material aspect of VIV amplification.
II. To conduct dynamic solution and UDF (Unit Defined Function) to obtain better observation on the amplification of vibration.
III. To conduct the experimental analysis for more reliable outcome.

REFERENCES

1. Gordon, A. L., R. D. Susanto, A. Ffield, B. A. Huber, W. Pranowo, and S. Wirasantosa (2008), Makassar Strait throughflow, 2004 to 2006, Geophys. Res. Lett., 35
.
2. Jiang, L. (2013). Deepwater Internal Wave Study and Application. M Sc Thesis, University of Texas.

3. Koto, J. & Negara, R. B. (2017). Potential of 100 kW of Ocean Thermal Energy Conversion in Karangkelong, Sulawesi Utara, Indonesia. International Journal of Environmental Research & Clean Energy, 5, 1-10.

4. Lienhard, I. H. (1966). Synopsis of lift, drag and vortex frequency data for rigid circular cylinders. Washington: Washington State University.

5. Susanto, R. D., Ffield, A., L.Gordon, A., & Adi, R. (2012). Variability of Indonesian throughflow within Makassar Strait,2004–2009. Journal Of Geophysical Research, 117.

6. Sumer, B. M. and Fredsøe, J. (1997). Hydrodynamics around cylindrical structures (Vol.12). World Scientific.

7. Syamsuddin, M. L., Attamimi, A., Nugraha, A. P., Gibran, S., Afifah, A. Q., & Oriana, N. (2015). OTEC Potential in the Indonesian Seas. Energy Procedia, 65, 215-222.