International Journal of Marine Engineering Innovation and Research, Vol. 8(4), Dec. 2023. 676-681
(pISSN: 2541-5972, eISSN: 2548-1479)

Analysis Of Magnus Effect Toward The Shaft Of Vetical Axis Hydro Turbine H-Darrieus

Irfan Syarif Arief[1], Amiadji[2], I Putu Gedhe Adhi Darsana[3] Achmad Baidowi[4]
(Received: 17 October 2022 / Revised: 24 January 2023 /Accepted: 23 November 2023)

***Abstract*— One way to fight climate change is making a transition from fossil fuel powered energy into renewable energy. In Indonesia the government have the national energy mix prediction which state that in 2050 58% Indonesian energy source will be renewable. The biggest source of renewable energy in Indonesia come from marine source. One sources of marine energy are tidal currents that can be harness by using hydrokinetic turbine. The goal to be solved is to determine the effect of magus force toward the turbin shaft. Shaft rotation speed and fluid velocity will be determined as the variation. The method used is computational fluid dynamic using fine marine numeca software to determine magnus force magnitude and mdsolids software to calculate the maximum bending moment after the magnus force applied. The results is magnus force just increase the minimum required diameter of turbine shaft by 0.26% and the corresponding safety factor is 1.889 more than 1.0 thus there is no need to replace or strengthen the existing turbine shaft..**

Keywords— *Hydrokinetic Turbine, Magnus Effect, Renewable Energy, Shaft Diameter*.

I. INTRODUCTION

One way to fight climate change that already happen right now is by making a transition from fossil fuel powered energy into renewable energy. The electricity generation sector contributes 25% of the world's greenhouse gas emissions and will continue to increase if there is no diversification of energy sources from fossil fuels. In contrast to emissions from electricity generation which need to be reduced ministry of energy states that the need for electrical energy in Indonesia actually increases 4.3% annually, coupled with government policies related to the national energy mix which show an increase in the use of new renewable energy which can reach up to 58% of total other sources by 2050to support the COP 26 contract Glasgow, Scotland October - November 2021 in which Indonesia stated that it would become a net zero emission country in 2060. The above adds to the long list of why new and renewable clean energy sources should be developed. Indonesia is a maritime area, with many seas, rivers and canals that have enormous potential to be utilized as a source of clean renewable energy, especially in hydropower energy sources which according to Koei, 2011 have a potential of 26,321 MW. Hydropower has become a renewable energy that uses water as its driving force and has been used for more than 16% of total power plants worldwide. One example of hydropower application is using a hydrokinetic turbine [1] [2]

Basically, hydrokinetic turbines are an adaptation of wind turbines where the technology used in hydrokinetic turbines is similar to wind turbines. Hydrokinetic turbine is an environmentally friendly technology. The electrical energy produced by this turbine can be utilized through the conversion of mechanical energy into electrical energy which must use the appropriate technology [3].

Hydrokinetic technology utilizes the flow of water to drive a turbine.Hydrokinetic turbines use a shaft to channel the kinetic energy of the fluid flow captured by the turbine blades to a generator so that it can produce electricity. The cylindrical axis which rotates at a certain rotational speed (RPM) is then exposed to a fluid flow with a certain speed (V m/s) which will cause a magnus effect that causes a force perpendicular to the direction of the fluid flow [2] [4].

One of the uses of the magnus effect is on the flettner rotor where a cylinder that stands on the ship's deck is rotated using an electric motor and when exposed to wind flow the cylinder will produce thrust which is perpendicular to the direction of the wind flow[5]

The magnus effect on the application of the flettner rotor on the ship is beneficial because it provides additional thrust to the ship, while the magnetic effect on the hydrokinetic turbine shaft is what adds to the force load on the shaft structure. In order to analyze the magnus effect, Computational Fluid Dynamic or CFD method is often chose to help researchers understand how it work. CFD is a numerical simulation method and data structures that analyze the characteristics condition that has been set. It helps the engineer to simulate a machine or products, so it doesn’t take time and high cost. In terms of this project, this method can analyze the fluid flow around the rotor and find several parameters desired.

II. METHOD

A. Literature Study

Irfan Syarif Arief, Department of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, 60111, Indonesia. E-mail: irfansya@its.ac.id
Amiadji Departement of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, 60111, Indonesia. E-mail: amiadji@its.ac.id
I Putu Gede Adhi Darsana, Departement of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, 60111, Indonesia. E-mail: b.adhidarsana@gmail.com
Achmad Baidowi, Departmen of Marine Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia. E-mail : ahmadbai@gmail.com

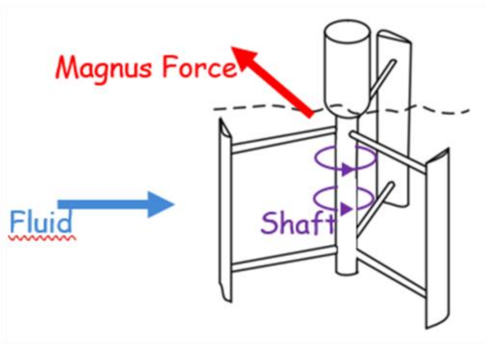

Figure. 1. Magnus Force in Turbine Shaft

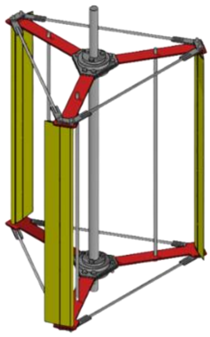

Figure. 2. Hydrokinetic Turbine

The first step of this project is to study the literature and gather the data that will be used in this project. The gathered data such as the shaft dimension, canal flow velocity, and turbine rotational speed is used for simulation input. This project will use a real case study that provided by hydrokinetic electricity generation project held by ITS Tekno Sains and Pembangkitan Jawa-Bali UP Paiton. The turbine dimension is two meters in height, diameter 1.7 meter with three blade and 3,000 watt capacity. The shaft use alloy steel type aisi 316 with 250 Mpa in yield strength. The shaft dimension is 2 meters in height and diameter is 0.06985 meter.

B. Modelling

The Modeling is done by making two main areas, first, the cylinder field, where the cylinder will become the shaft which is then calculate the value of magnus force. Then the second is the domain field, the field is useful as fluid medium or fluid flow. Then the two fields are defined boundary condition, which determines the fluid input, fluid output, solid plane, and external field. If it has been done then it can be continued with the meshing process. Because of the uncomplicated shape of the flettner rotor or just a cylinder, the determination of the meshing density does not need to be too high. Because if made high density then consequently the process of running or simulation takes longer with results that are not much different if made with a simple density level.

The model of shaft that will be used in running the simulation is a plain cylinder with 0.03495 meters radius and 2 meters height. Numeca Hexpress are used to designing the three dimensional model of the shaft.

C. Simulation Process

Pre- Processor

Consist of the input from a flow problem that will be simulated. Defining domain, fluid properties, and boundary condition is located in this step. The object to be analysed is also going through meshing process in this step.

Then input parameters, where variations of RPM flettner rotor and fluid velocity can be performed. In addition, other values can also be included, like the type of fluid, flow type, and so forth. In this simulation the fluid flow direction is assumed to flow to x positive axis on cartesian coordinat based on the model.

Running process is done 51 times with 5 variation of fluid velocity i.e.1.2 m/s; 1.6 m/s; 2.0 m/s; 2.4 m/s; 2.9 m/s. and 10 variation of shaft rotational speed 12,17, 22, 32, 42, 52, 62, 72, 77, 80 RPM plus a simulation without shaft rotation.

Processor

In solver step, include the stage of calculation for every iteration in parts associated with the mesh configuration and the parameters/ method being inputted. The processing or running can be monitored from monitor menu.

D. Magnus Force Calculation and Its Effect Toward The Turbine Shaft

Several formulas are used to analyse the magnitude of magnus force that are provided by the simulation result. The simulation result are unit of force Fx: force that is unidirectional with the fluid flow and Fy: force that is perpendicular to the direction of flow.Taguchi method is a methods used in off line quality control activities at the design process stage of production, what is meant by off line quality control is quality control activities in product design. In the Taguchi method a matrix called orthogonal array is used to determine the number of minimal experiments that can give information as much as possible of all factors that influence the parameters. The most important part of orthogonal arrays lies in the selection of combinations level of input variables for each experiment.

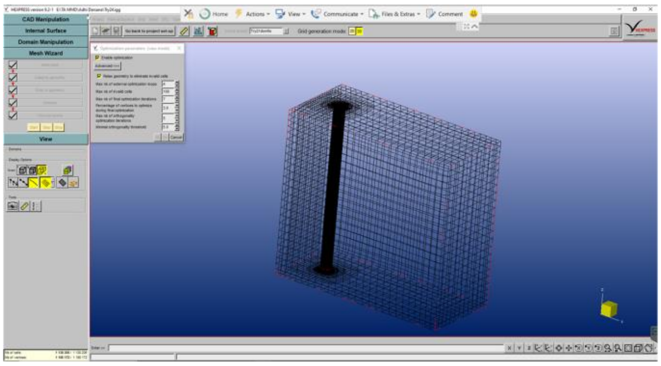

Figure. 3. Magnus Force in Turbine Shaft

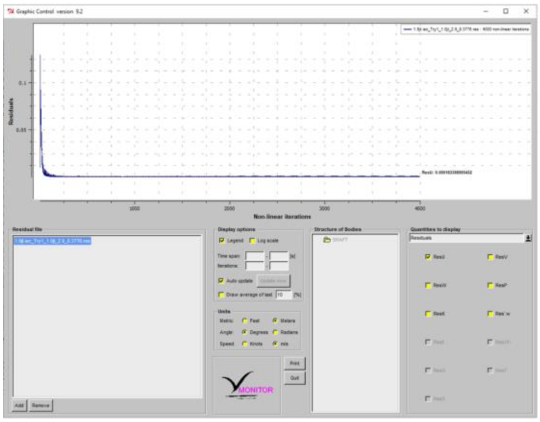

Figure. 4. Hydrokinetic Turbine

Post Processor

Post processing is an end step process which consist of result from the calculation. Visualization from the formulation is also can be seen in this step with variation of 2 dimension or 3-dimensional view [6].

TABLE 1.
TURBINE SHAFT DIMENSION

Turbine Shaft	
Diameter	0.06985 m
Height	2 m
Aspect Ratio (H/D)	28.63

TABLE 2.
SIMULATION INPUT PARAMETER

Parameter	Definition
General Parameter	Unsteady
Fluid Model	Salt Water 25^{0}C, 0.00925Pa.s Dynamic Viscosity, 1023.3878kg/m3 Density
Flow Model	EASM Reference Length: Diameter Poros Reference Velocity: Variasi Kecepatan Fluida
Reference Velocity	Variasi Kecepatan Fluida
Boundary Condition	Solid: Wall-function, External: Far field velocity Vx(Wind speed variance)
Body Definition	SHAFT
Body Motion	RZ0 (yaw) Imposed (Variasi RPM)
Mesh Management	Activate Frame Rotating Method
Initial Velocity	Variasi Kecepatan Fluida

Aspect Ratio

The AR represents the ratio between height and diameter. Thus, from this ratio the value of shaft's height ...er value is ... he

$$Re = \frac{u\ (^{m}/_{s})\ x\ L\ (m)}{V\ (m^2/s)} = \frac{\rho\ \left(^{kg}/_{m^3}\right)\ x\ u\ (^{m}/_{s})\ x\ L\ (m)}{\mu\ (Pa.s)} \quad (5)$$

$$AR = \frac{H\ (m)}{d\ (m)} \quad (1)$$

Spin Ratio`

It is another factor that influence the aerodynamic characteristics. It represents the ratio between the circumferential speed of the shaft and the free stream velocity. Based on the graph, the SR value chosen for this project is 3 and the value of the wind speed is to be varied from 7.2 meter/second until 2.7 meter/second. Therefore, the value of rotor's RPM can be determined and varied as well. When the AR and SR value are identified, the next parameter to be decided is the end plate.

$$SR = \frac{\Omega\ \left(\frac{rad}{s}\right) \times d(m)}{2\ \times\ u\ (m)} \quad (2)$$

Lift and Drag Force

Lift is a force that is perpendicular to the oncoming flow direction thus this use Fy value. It is created by an object that rotating through a moving fluid resulting in a pressure difference among the body and start to create a lift force. While drag is a force that acts opposite to the relative motion of an object moving with respect to a surrounding fluid thus assume this use Fx value. Sometimes it is called as an air resistance or type of friction. These forces will used as a reference for the force that the shaft will produce. [7], [8]

$$CL = \frac{2\ x\ L\ (N)}{\rho\ \left(^{kg}/_{m^3}\right) x\ u^2\ (^{m}/_{s})\ x\ S\ (m^2)} \quad (3)$$

$$CD = \frac{2\ x\ Fd\ (N)}{\rho\ \left(^{kg}/_{m^3}\right) x\ u^2\ (^{m}/_{s})\ x\ S\ (m^2)} \quad (4)$$

Reynold Number

The Reynolds number describes the relationship between inertial and viscous forces within a fluid that is experiencing relative internal motion as a result of varying fluid velocities used as a reference for the force that the shaft will produce [9].

Shaft Diameter

Shaft diameter is function of bending moment and torsion. Moment is affected by the forces acting in shaft, because of the magnus force the force acting in turbine shaft will increased. The effect of increased force toward the shaft diameter can be calculated with this formula [10].

$$D^3 \geq \frac{N\ x\ 16}{\pi\ x\ 0.5\ x\ \tau}\ x\ \sqrt{(M^2 + T^2)} \quad (4)$$

Safety Factor of Shaft

To know the shaft afte effected by magnus force is still capable to withstand the force acting on it, safety factor will be calculated using two method manual calculation from equation no.4 and numerical simulation

IV. RESULTS AND DISSCUSION

The result of Numeca Fine Marine Software can be viewed by selecting the CFView menu for viewing the contour of simulation output like velocity and pressure.

To see the distribution of pressure occurring around the fluid can select the static pressure menu in the quantities dialog box. In the picture shows that there is a large pressure difference between the two sides of the shaft. Where the reddish color signifies high pressure while the blue color tends to be the opposite. From the whole simulation performed the pressure distribution that happened tend to same. Because on the whole simulation effect magnus work.

Based on Bernoulli's law the value of pressure will be inversely proportional to the value of velocity. So if we take the picture the distribution of the fluid velocity around the shaft will be seen in contrast to the image at the pressure distribution. See figure 6 and figure 7.

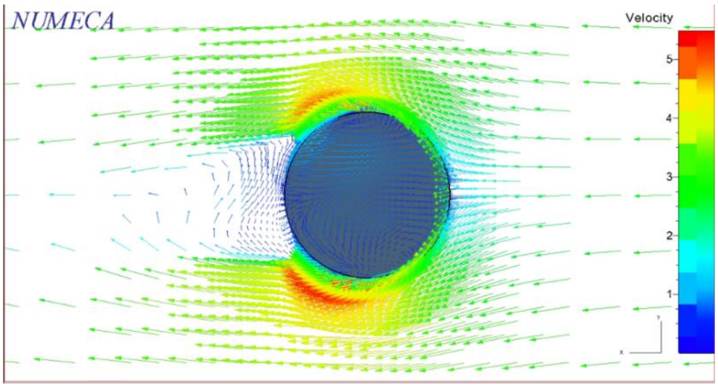

Figure. 5. Velocity Contour

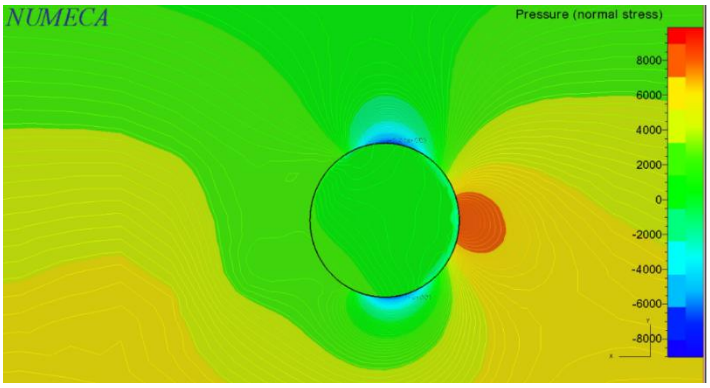

Figure. 6. Pressure Contour

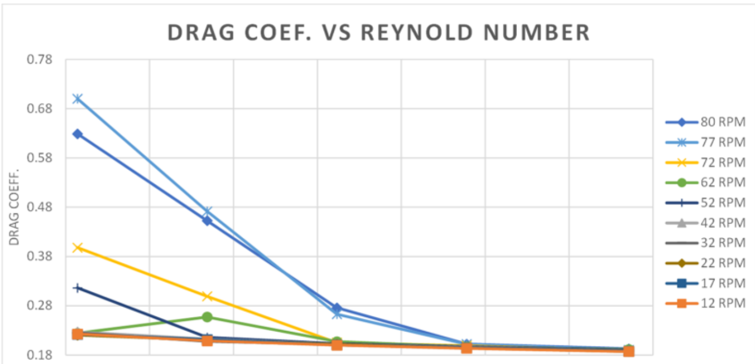

Figure. 7. Cd and Reynold Number

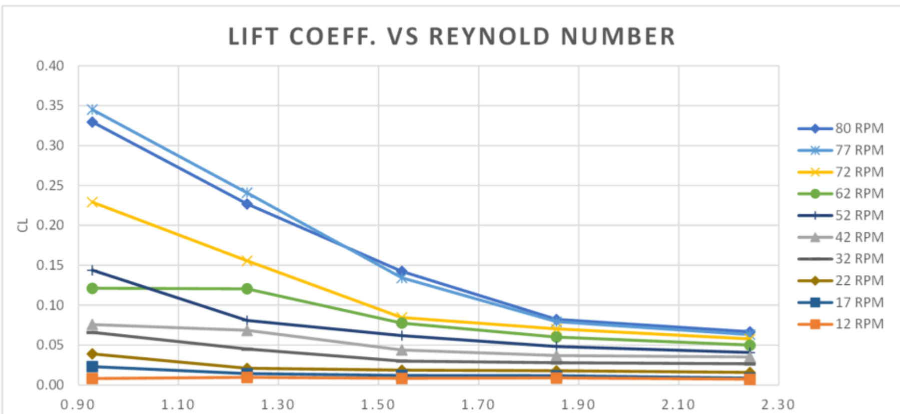

Figure. 8. CL vs Reynold Number

For the relation of magnus force and shaft diameter see figure.

The increasing diameter does not reach the existing diameter. Minimum Safety factor are 1.0. The Manual calculation result is 1.9041 without magnus and 1.8894 with maximum magnus force, and simulation result for safety factor are 1.9647 without magnus and 1.9590 with maximum magnus force.

V. CONCLUSION

The higher the Reynolds number, the lower the lift and drag coefficient produced. This shows that the Reynolds number is inversely proportional to the lift coefficient. Meanwhile, the higher the rpm, the higher the lift coefficient.

The higher the spin ratio, the higher the lift and drag coefficient produced. This shows that the spin ratio is directly proportional to the lift coefficient.

From the calculation and simulation of safety factor the value still meets the minimum safety factor requirements of 1.0 thus the existing turbine shaft does not need to be replaced or strengthened.

ACKNOWNLEDGMENT

This research was supported by MMD Laboratory, and Department of Marine Engineering, Institut Teknologi Sepuluh Nopember Surabaya. I dedicated this whole research to all of you, to my family, and my friends who always supported me during finishing this paper.

REFERENCES

[1] Utomo, Cahyo. 2017. *Perancangan dan Pembuatan Dies Permanent Mold Pengecoran Logam Dengan Material Besi Cor Ductile*. Surakarta: Universitas Muhammadiyah Surakarta

[2] Firdaus. 2009. *Rancang Bangun Cetakan Permanen (Permanent Mold) Untuk Pembuatan Pulley Aluminium*. Palembang: Politeknik Negeri Sriwijaya

[3] I G.N.K. Yudhyadi, T. R. 2016. *Optimasi Parameter Permesinan Terhadap Waktu Proses Pada Pemrograman CNC Milling dengan Berbasis CAD/CAM*. Dinamika Teknik Mesin, VI (1), 38-50.

[4] Rochim, Taufiq. 2002. *Optimasi Proses Pemesinan*. Bandung: Institut Teknologi Bandung

[5] Supriyanta. 2018. *Desain Dan Proses Pembuatan Cetakan Permanen Dengan Material Logam Besi Cor Kelabu Hasil Coran Pasir Co2 Untuk Proses Pembuatan Flange Dengan Material Kuningan*. Surakarta: Universitas Muhammadiyah Surakarta.

[6] Adi Nugraha, Sasmita. 2018. *Optimasi Desain Propeller Sea Water Pump Dengan Parameter Distribusi Blade Angle Pada Daerah Shroud Menggunakan Pendekatan Computational Fluid Dynamics (CFD).* Surabaya: Institut Teknologi Sepuluh Nopember

[7] Wijanarka, B. 2013. *Cadcam untuk Mesin Bubut dan Frais Menggunakan MasterCam 9 dan X3*. Yogyakarta.

[8] Sumbodo, W. 2008. *Teknik Produksi Mesin Industri Jilid 1*. Jakarta. Direktorat Pembinaan Sekolah Menengah Kejuruan

[9] Kumbhar. 2015. *Multi-objective Optimization of Machining Parameters in CNC End Milling of Stainless Steel 304*. India: K.B.P. College of Engineering

[10] Atmadio, Nico. 2018. *Optimasi Jalan Pahat dan Analisis Biaya Produksi Proses Permesinan CNC Lathe Pembuatan Piston Master Cylinder Rem Sepeda Motor Yamaha Menggunakan CAM*. Pekanbaru: Universitas Riau

[11] Daniel. (2009). *Optimasi Parameter Pemesinan Proses Cnc Frais Terhadap Hasil Kekasaran Permukaan Dan Keausan Pahat Menggunakan Metode Taguchi.* Semarang: Universitas Diponegoro

[12] Bagus Praetyo, A. 2015. *Aplikasi Metode Taguchi Pada Optimasi Parameter Permesinan Terhadap Kekasaran Permukaan Dan Keausan Pahat Hss Pada Proses Bubut Material St 37*. Kediri: Universitas Nusantara Persatuan Guru Republik Indonesia

[13] Hamdani, Ficky. 2014. *Optimasi Pemesinan Pada Mesin Bubut Tipe M-300 Horrison Dengan Metode Optimasi Algoritma Genetika*. Medan: Universitas Sumatera Utara

[14] Candra, Susila. 2009. *Optimasi Proses Pemesinan Milling Fitur Pocket Material Baja Karbon Rendah Menggunakan Response Surface Methodology*. Surabaya: Universitas Surabaya

[15] Anto, Edi. 2013. *Optimasi Parameter Permesinan Terhadap Waktu Proses Pada Pemrograman CNC Turning*. Semarang: Universitas Negeri Semarang

[16] Iskandar, W. 2016. *Analisa Teoritis Kebutuhan Daya Mesin Bubut Gear Head Turret*. Surakarta: Universitas Muhammadiyah Surakarta.

[17] Mosey, C.A. 2015. *Perhitungan Waktu Dan Biaya Pada Proses Permesinan Benda Uji Tarik*. Manado: Universitas Sam Ratulangi

[18] Ibnu Sabilillah, M. 2018. *Perancangan dan Perhitungan Waktu Total Proses Produksi Blade Controllable Pitch Propeller Dengan CNC untuk Perencanaan Biaya Produksi*. Surabya: Institut Teknologi Sepuluh Nopember.

[19] Van Hoten, Henry. 2012. *Pemodelan Sistem Perhitungan Tarif Mesin Produksi Dengan Memperhitungkan Beban Langsung dan Tak Langsung*. Bandung: Institut Teknologi Bandung.