EMITTER International Journal of Engineering Technology
Vol. 6, No.1, June 2018
ISSN: 2443-1168

Improve of Water Flow Acceleration in Darrieus Turbine Using Diffuser NACA 11414 2,5R

Setyo Nugroho, Arrad Ghani Safitra, Teguh Hady Aribowo, Mochammad Arief Julianto

Power Plant Engineering, Electronic Engineering Polytechnic Institute of Surabaya
Jalan Raya ITS Sukolilo, Surabaya 60111
setyo@pens.ac.id, arradgs@pens.ac.id, teguhhady@pens.ac.id

Abstract

Indonesia has potential hydro energy around 70000 MW which has been used around 6% (3529 MW). One of the development constraint is the stream velocity in Indonesian rivers is relativey low. It causes bigger turbine dimension needed to achieve power which is desired. An alternative is to utilize a diffuser, which is a device that could accelerate the fluid flow in order to give more energy to the turbine. Based on contiunity equation, diffuser can increase velocity by ratio of cross-section area. It can be used to achieve expected power as long as it is not too much reduce the pressure. This research was conducted in 0.566 m/s of water velocity with Darrieus turbine with hydrofoil NACA 0018, height 0.74 m, radius 0.17 m, chord 0.11 m and 3 number of blades. The performance (Cp) was determined by numerical and experimental without and with diffuser NACA 11414 2.5R for variation of angle 8^o, 16^o, and 20^o. Both of those results showed that the best performance of NACA 11414 2,5R is on angle 16^o which numerically has stream velocity 0,91 m/s of water and 7 times of Cp, while experimentally has 0,891 m/s of water velocity and 3,16 times of Cp. This diffuser could improve the power generated by the turbine and increase the turbine efficiency.

Keywords: Darrieus water turbine, diffuser, coefficient of performance.

1. INTRODUCTION

Development of renewable energies is one of efforts to decrease dependence of fossil energy and its impact on global warming. Indonesia has potential hydro energy around 70000 MW which has been used around 6 % (3529 MW). Hydro power plant usually need high head for generating potential energy that could be converted to kinetic energy and finally to electricity. One method to harness the hydro energy is using vertical axis turbine, like Darrieus turbine. It is a method which no need high head, but could be enough to convert the energy kinetic of water to energy mechanic to rotate generator. The Darrieus turbine need less components and lower cost and can implimented on outlying area.

Velocity of river whose is relatively low become challenge for implementing a vertical axis turbin. An example is Kromong river, Pacet where this research was conducted has water velocity around 0.56 – 0.76 m/s. The solution is adding diffuser on turbine system in order to increase the performance of the turbine. Based on the contiunity equation, diffuser can increase water velocity by ratio of cross-section area. Its can used to increase generated power. The previous work conducted a diffuser with chambered surface and NACA 11414 and Eppler 420 with narrow range angle, therefore this research used NACA 11414 with wider range angle in order to reduce the pressure loss. This research was conducted in 0.566 m/s of water velocity with Darrieus turbine, hydrofoil NACA 0018, height 0.74 m, radius 0.17 m, chord 0.11 m and 3 number of blades. Then the calculation of the performance (Cp) was done by numerical and experimental method ;without and with diffuser NACA 11414 2.5R for various of angle 8^o, 16^o and 20^o. Both of those results showed that the best performance of NACA 11414 2.5R is on angle 16^o whose numerically has 0.91 m/s of water and 7 times of Cp while experimentally has 0.891 m/s of water velocity and 3.16 times of Cp.

2. RELATED WORKS

Velocity Increasement Due To Diffuser

In previous work, there were researches which study of velocity increasment due to the diffuser. The velocity coud increase to 30% [1] due to the chambered surface of diffuser. This chambered surface will reduce the pressure of the stream as shown in Figure 1.

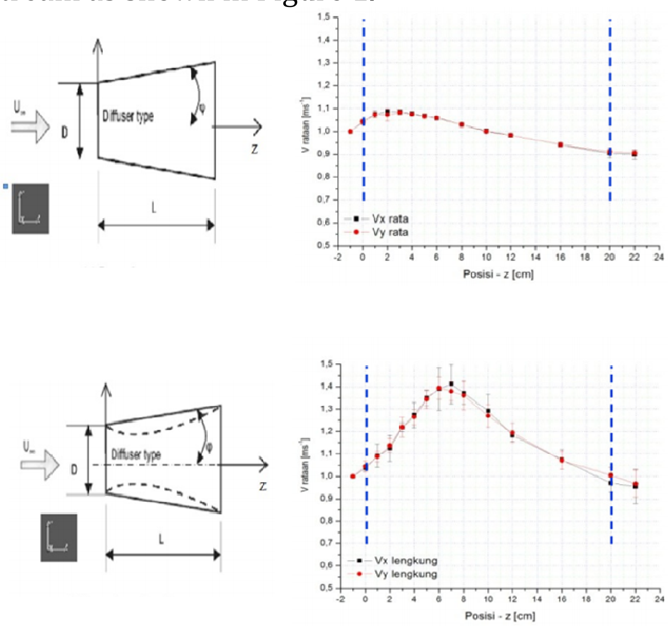


Figure 1. Velocity distribution of the stream along the diffuser

Those results showed that a shape and angle of diffuser could increase the flow velocity. Then this research investigated the optimum angle which produce the largest incerement of velocity.

Performance Increasement of Darrieus Turbine

In previous work, there were researches which analyze the influence of diffuser in turbine performance [2]. In various shapes of the diffuser showed that the diffuser could increase the turbine performance. The performance could increase 2 – 4 times experimentally, and 5 - 15 times numerically as showed in figure 2.

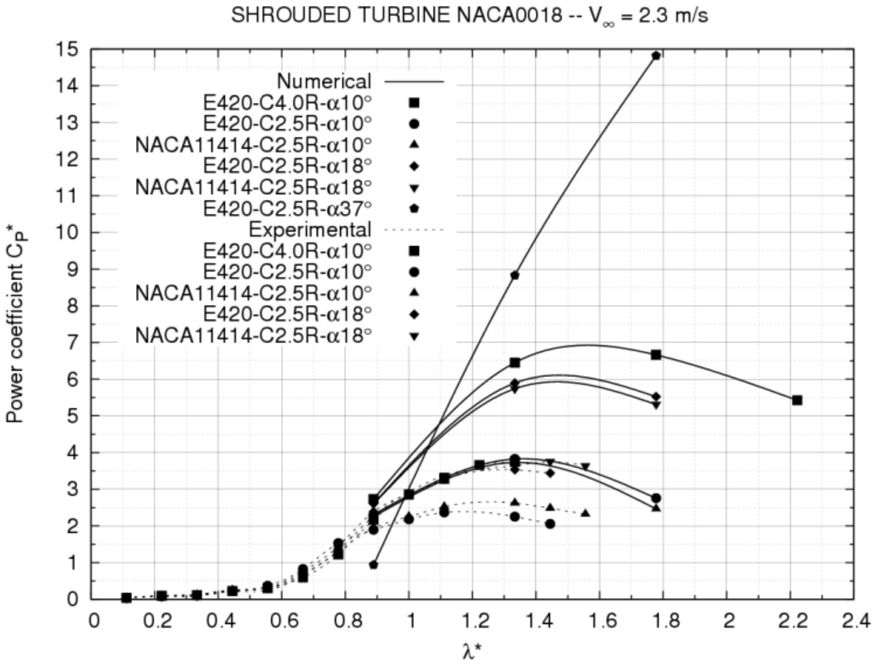
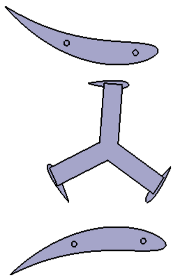


Figure 2. Darrieus turbine performance with various diffusers

Then the velocity which had increased was improve the performance of Darrieus turbine significantly as shown in figure 2. The best result was obtained with diffuser using NACA 11414 with 18^0. The reason was because the diffuser will accelerate the flow and could make less torque fluctuation acted on the blade. So, this research used the same NACA 11414 and investigated the variation around 18^0 in order to know the better result than before.

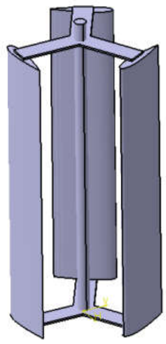
3. SYSTEM DESIGN

The research was conducted to look at the performance increasement of Darrieus turbine due to the diffuser. The diffuser using airfoil NACA 14114 2.5R, and the performance evaluated due to variations of diffuser angle. The research conducted in 3 variations diffuser angle; 8^o, 16^o, and 20^o. Finally the maximum performance achieved in certain diffuser angle for certain TSR. Figure 3 show the full specification of the diffuser, and Figure 4 show the specification of Darrieus Turbine.



Diffuser Specification	
Hydrofoil	NACA 14114
Chord	2,5 R
Span	0,8 m
Angle	8°, 16°, 20°

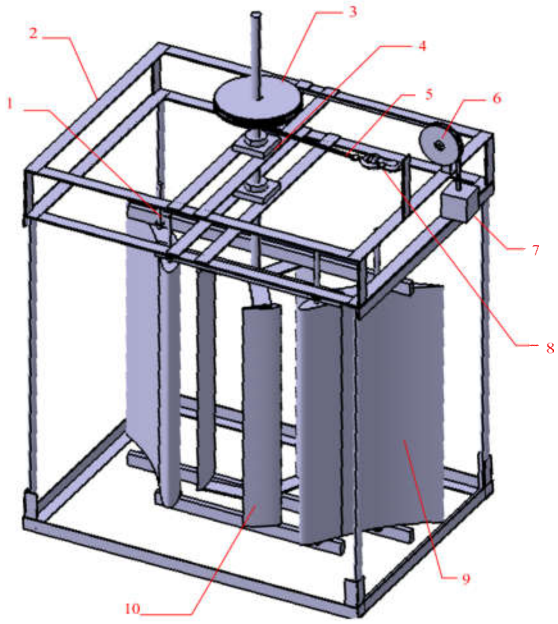
Figure 3. Diffuser specification



Darrieus Turbine Specification	
Hydrofoil	NACA 0018
Chord	0,11 m
Diameter	0,34 m
Span	0,74 m
Solidity	1,6
Shaft Diameter	1,5 inch

Figure 4. Darrieus turbine specification

The turbine and the diffuser assemblied as shown in Figure 5.



Nomenclature :

1. Slider	3. Pulley	5. Rope	7. Load	9. Diffuser
2.Turbine Housing	4. Bearing	6. Katrol	8. Load Scale	10. Turbine

Figure 5. System design

This system was designed to measure the torque generated by rotation of the turbine due to water stream. The torque measured by a load scale with the rope brake principal. This principal is by giving a frictional load the rotating shaft. This frictional load was generated on a rope by a balancing load. When the turbine is not rotating, the rope is tense due to balancing load. Then when the turbine is rotating, the additional load due to inertial momen of the turbine will be generated. This additional load could be interpreted as the torque by this following relation.

$$\begin{aligned} T &= F \cdot r \\ &= W \cdot r \\ &= \Delta m \cdot g \cdot r \end{aligned} \tag{1}$$

where, T is torque (N.m)
F is balancing load (N)
r is pulley radius (m)
W is load (N)
Δm is additional mass (Kg)
g is gravitional acceleration (9,81 m/s^2)

Then the rotational speed (ω) was measured by digital laser tachometer, so now the mechanical power could be determined by

$$P_{diffuser} = T \cdot \omega \tag{2}$$

There is a better method to measure the torque which use dynamic torque sensor. That method will give a better accuracy, but unfortunately this method not conducted in this time because of the sensor was unavailable.

The flow velocity itself was measured by currentmeter as shown in figure 6. The measurement was performed in the way upstream and in the middle between the diffuser just before the flow hit the turbine.

Figure 6. Cup Current Meter

with current flow velocity and rotational velocity, The Tip Speed Ratio could be calculated by

$$TSR = \frac{\omega_d \cdot r}{V} \quad (3)$$

ω_d is rotational speed of turbine

r is turbine radius

V is flow velocity

4. NUMERICAL SIMULATION

The experimental and numerical was perfomed in this research. Experiment was conducted in Kromong river, Pacet. Numerical study was conducted in Computer Laboratory, Power Generation System, Electronic Engineering Polytechnic Institute of Surabaya.

The numerical method simulated the velocity increment and turbine performance increment. The stream velocity increment simulated by Computational Fluid Dynamic (CFD) method which was commonly used to predict the outcome of a research.

In this research, the simulation of diffuser used two dimensional model with actual operating geometry of the diffuser. The simulation parameters was listed in Table 1, and the simulation domain an meshing was shown in Figure 6 dan Figure 7. The simulation parameter and method refered to [3], [9]

Table 1. Simulation modelling method

Models	
Viscous	: Realize k-Ɛ
Species	: Species Tansport – No Reaction
Boundary conditions	
Inlet	: Velocity inlet
Diffuser	: Not moving wall
Outlet	: Outflow
Solution Methods	
Pressure Velocity Coupling	: Simple
Pressure	: Standart
Momentum	: First Order Upwind
Turbulent Kinetic Energy	: First Order Upwind
Turbulent Dissipation Rate	: First Order Upwind

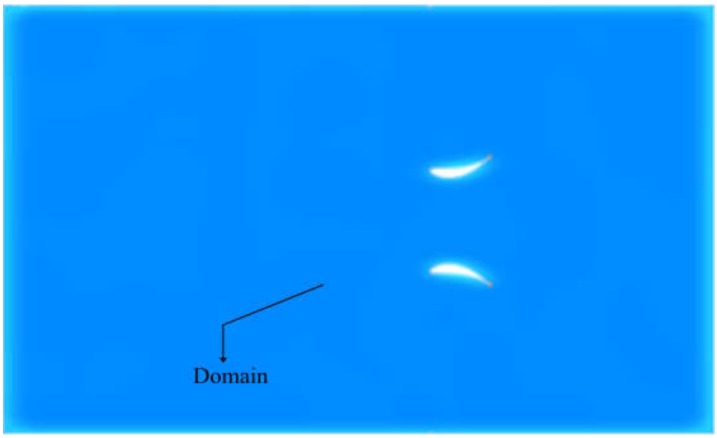
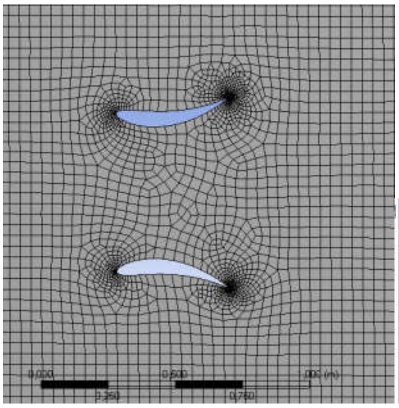


Figure 7. Simulation domain



Statistics	
Nodes	9909
Elements	9647
Mesh Metric	Orthogonal Quality
Min	0,704286341712911
Max	1
Average	0,989725554345208
Standard Deviation	2,78105205544057E-02

Figure 8. Simulation meshing

5. RESULTS AND ANALYSIS

The performance of the turbine was determined in two aspects. The first one is due to the effectiveness of the diffuser which means the diffuser could accelerate the freestream velocity to maximize the turbin performance. This effectiveness was evaluated by Computational Fluid Dynamic (CFD) method which predicted the acceleration due to the diffuser and then the resulls compared with the experimental datas. The second one is the increasement of turbine performance itself. The turbine performance could increase because of the freestream velocity was increased by the diffuser, so the power generated by the turbine rose. The turbine performance evaluated numerically by Double Multiple Streamtube (DMST) method and the results compared with the experimental datas.

Diffuser Performance

The performance of diffuser NACA 11414 2,5R 8^o, 16^o dan 20^o angles could be determined with equation (4) to (6) [3], [4]

$$V_1 = \beta \cdot \gamma \cdot V_0 \tag{4}$$

$$\beta = \frac{x_3}{x_1} \quad (5)$$

$$\gamma = \frac{V_3}{V_0} \quad (6)$$

Where V_0 is the freestream velocity which is 0.566 m/s and the others parameters explained in Figure 9

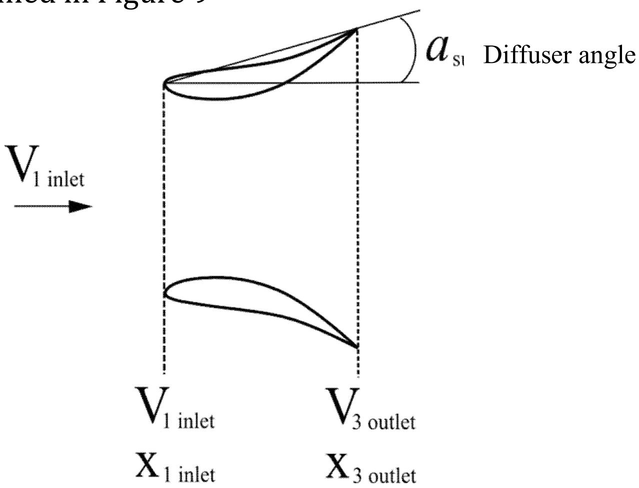


Figure 9. Diffuser nomenclature

The free flow velocity in experiment was measured by currentmeter, and by user input in numerical simuationAnd the results shown in Table 2.

Table 2. Performance of diffuser on every angle

Parameter	Simulation			Experiment		
	8°	16°	20°	8°	16°	20°
V_1	0,784 m/s	0,891 m/s	0,825 m/s	0,72 m/s	0,91 m/s	0,87 m/s
V_3	0,62 m/s	0,569 m/s	0,51 m/s	0,77m/s	1,28 m/s	0,92 m/s
β	1,1	1,205	1,339	1,339	1,205	1,1
γ	1,095	1,005	0,883	1,36	2,261	1,625
βγ	1,2045	1,211	1,182	1,821	2,725	1,788

As shown in table 2, the maximum velocity inlet at the diffuser was occurring in 16° angle which is 0.891 m/s. The β values which means a ratio between outlet cross sectional area and inlet cross sectional area is increase in the larger angle, otherwise the γ factor which means the ratio between outlet velocity and free stream velocity is decrease in larger angle. The diffuser performance itself was quantified by β γ factor. The maximum performace achieved at the maximum β γ. In this case, the maximum performance was achieved in 16° diffuser angle which the acceleration of the flow reach the maximum value.

With the acceleration of the velocity gave additional power to the turbine. The power available by the stream and the increasement shown in Table 3.

Table 3. Power Availability

Angle	Numeric		Experiment	
	P	P*	P	P*
without	22,78	-	22,78	-
8°	60,56	2,65	46,95	2,06
16°	88,89	3,9	94,79	4,16
20°	70,56	3,09	82,83	3,63

The goal of this research is to know the incensement of turbine performance with and without the diffuser. In Table 3 shows that the increasement of the power available

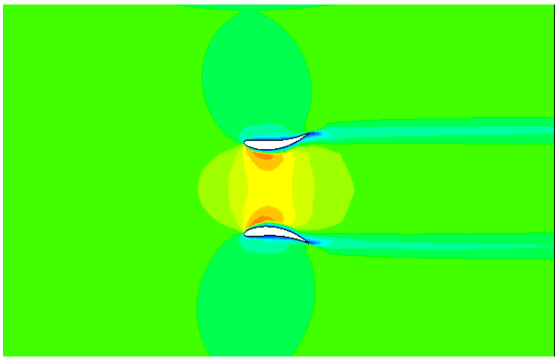
$$P^* = P_{difuser} / P_{without\ diffuser} \tag{7}$$

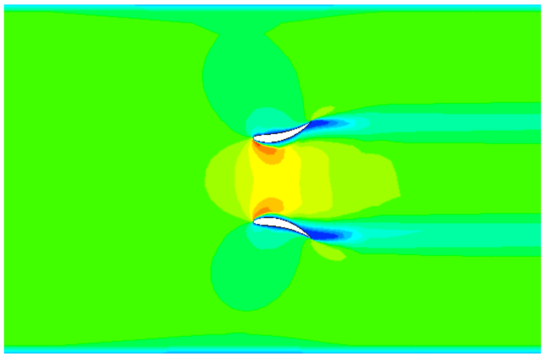
could reach 4.37 at the 16° diffuser angle, where the power (P) itself could be determined with equation (8)

$$P_{avail} = \frac{1}{2}\rho \cdot A \cdot V^3 \tag{8}$$

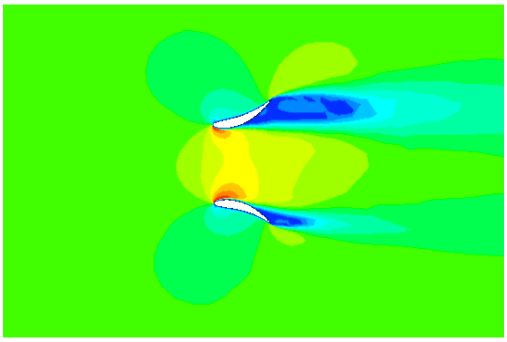
This equation also explain that the most influencing factor to the turbine power is velocity. Velocity improvement on using diffuser directly provided more power availability for converted to mechanic power by turbine. By more power availability, turbine can produce more mechanic power. Table 2 presented improvement of power availability on the every diffuser angle. Highest power on diffuser 16°, that's because of the higest velocity on this angle. The improvement up to 3,9 times numerically and 4,16 times experimentally.

CFD simulation can explain how the flow behaves around the diffuser. Figure 10 and 11 show that the velocity was accelerated due to the reduction of cross sectional area in diffuser inlet, and the the flow decelerated as the increase of the cross sectional area along the streamwise. The maximum velocity occurred in the throat of the diffuser, so this is the place where the turbine located to gain the maximum power.

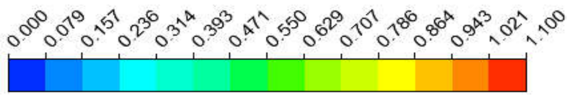




(a) (b)



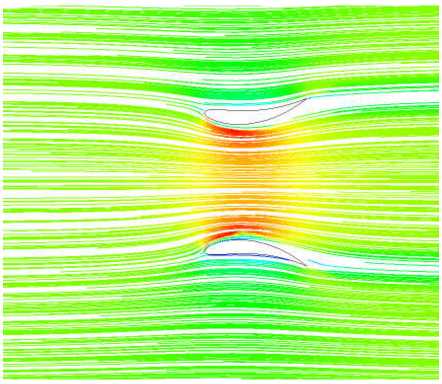
(c)

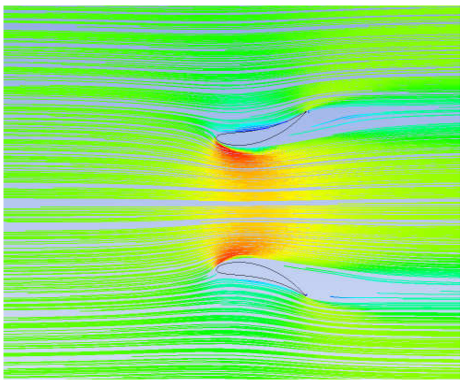


Velocity (m/s)

Figure 10. Velocity contour at the various diffuser angles (a) 8° (b) 16° (c) 20°

As shown in figure 10, there is pressure drop behind the guide vane larger as the increase of the angle. This pressure drop is undesired by the turbine because it will reduce the turbine performance. Therefore an angle position should consider the acceleration of the flow and the pressure drop occurred. From the results, the best angle is 16° which give a good acceleration with acceptable pressure drop.





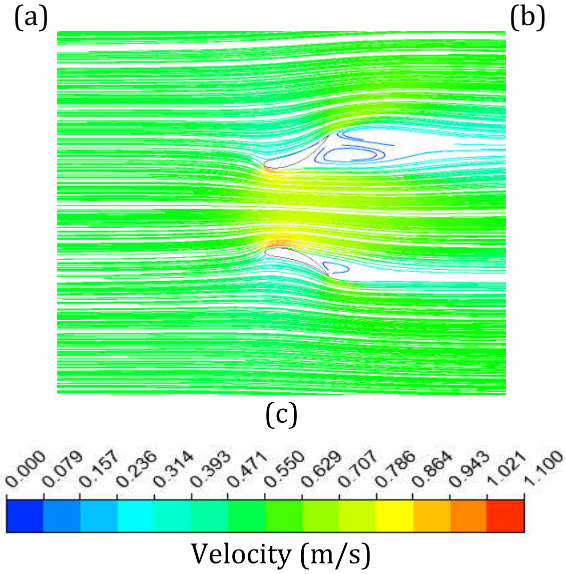


Figure 11. Streamline water velocity at various diffusre angle (a) 8° (b) 16° (c) 20°

Turbine Performance

Figure 12 presented diffuser on every angle can improved Cp and TSR of Darrieus turbine. Diffuser 16° experiences highest improvement, is 7 times numerically anda 3,16 times experimentally. It caused diffuser 16° provided more power availability compared diffuser 8° and 20°. Obvisiously, mechanic power is improved by more power avaibility. This is because in 16^0 the flow velocity accelerated with the maximum value

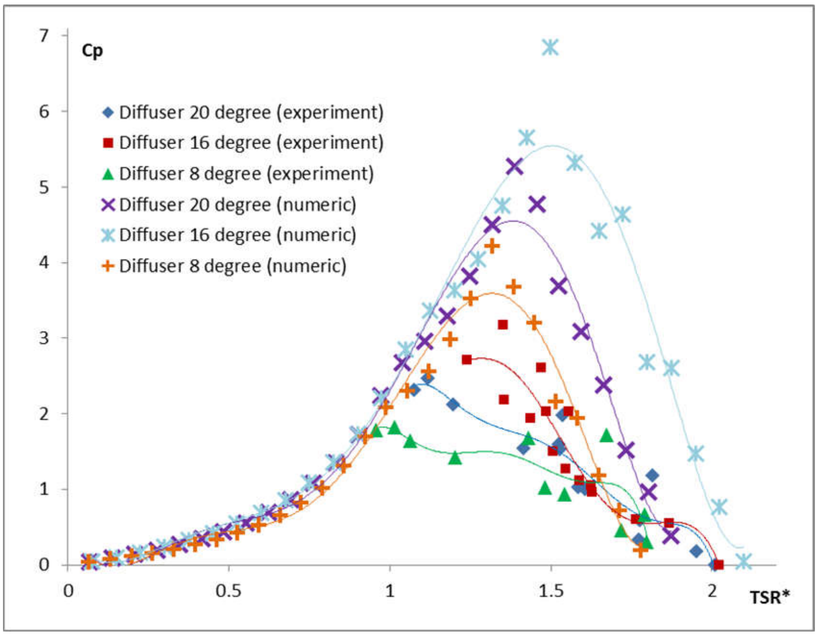


Figure 12. Relation Cp* and TSR*

The performance improvement was quantified by

$$Cp^* = \frac{Cp}{Cp_{opt}} = \frac{Cp}{0{,}28} \tag{9}$$

and

$$TSR^* = \frac{TSR}{TSR_{opt}} = \frac{TSR}{2{,}1} \tag{10}$$

Which explain the improvement performance by adding the diffuser compared with without one in specific TSR. This result was obtained with maximum 15% accuracy due to the constrain of the experimental setup, especially the torque measurement method.

6. CONCLUSION

This research showed that the applied diffuser could increase the turbine performance. Diffuser NACA 11414 2,5R with angle 16^o can increase coefficient of performance is 7 times numerically and 3,16 times experimentally. This is because the usage of the diffuser could accelerate the flow more effective than the usage of chambered surface or 10^0 diffuser. This accelerated flow increased the velocity used by the turbine. In future works should consider the adaptive angle of the diffuser in order to maintain the flow velocity due to fluctuative river flow.

Acknowledgements

Thanks to Fluid Laboratory and PLC Laboratory in Power Generation System at Electronic Engineering Polytechnic Institute of Surabaya that has provided the testing facilities so that this research could be done.

REFERENCES

[1] Deka W, Purwanto, Interior Lengkung Diffuser Untuk Peningkatan Performansi Diffuser Augmented Wind Turbine, Master Thesis, ITS (Surabaya), 2010

[2] Roa, A.M, "Numerical And Experimental Analysis Of A Darrieus Tipe Crossflow Water in Bared And Shrouded Configurations", IOP Conference series: Earth and Environmental Science Vol. 12, No. 1, 2013

[3] Khunthongjan, Palapum,"A Study of Diffuser Angle Effect on Ducted Water Current Turbine Performance Using CFD", Songklanakarin Journal Science and Technology, 2012

[4] I. Paraschivoiu, "Double-Multiple Stream Tube Model for Studying Vertical-Axis Wind Turbines," Journal of Propulsion and Power, Vol. 4, No.4, p.370. doi:10.2514/3.23076, 1988

[5] Brian, Kirke, ”Development in ducted water current turbines”, Tidal Paper (2005).page 1-12
[6] Maître Thierry, Mentxaka Ane, Pellone Christian,Jean-Luc,” Numerical 2D Hydrodynamic Optimization Of Channeling Devices For Cross-Flow Water Turbines”, U.P.B. Sci. Bull., Series D, Vol. 72, Iss. 1, 2010
[7] Mulkan Ibnu, Hantoro Ridho, Nugroho Gunawan, “Analisa Performansi Turbin Arus Sungai Vertikal Aksis Terhadap Penambahan Variasi Panjang Rasio Luasan Pada Diffuser”, JURNAL TEKNIK POMITS Vol.1, 2012
[8] Reinecke, Josh, “Effect Of An Diffuser On The Performance Of An Ocean Current Turbine”, Master Thesis University of Stellenbosch, 2011
[9] Setyo Nugroho, Ciptananda Citrahardani “CFD Analysis of Nozzle Exit Position Effect in Ejector Gas Removal System in Geothermal Power Plant", EMITTER International Journal of Engineering Technology, 2015