VOL. 15, NO. 22, NOVEMBER 2020 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

CROSS FLOW KINETIC TURBINE PERFORMANCE USING COMPUTER FLUID DYNAMIC SIMULATION

Rudy Soenoko and Erwin Sulistyo
Department of Mechanical Engineering, Faculty of Engineering, Brawijaya University, Malang, Indonesia
E-Mail: rudysoen@ub.ac.id

ABSTRACT

The purpose of this research is to observe a kinetic turbine with a new design. Kinetic turbine with this new design adopts a cross flow turbine design. As with kinetic turbines in general, kinetic turbines with this new design also only rely on the water flow rate to drive the turbine, which is then converted into electrical energy by utilizing a generator. It is hoped that this Cross Flow Kinetic Turbine can improve the performance of kinetic turbines, which are known to have a low efficiency. The research system conducted is a simulation that utilizes CFD software. This simulation activity is comparing water behavior in a kinetic turbine by observing the water trajectory between the turbine blades. The kinetic turbine that will be compared with the cross flow kinetic turbine (CFKT) is a Curve Bladed Kinetic Turbine (CBKT). The reason for comparing with CBKT is because the CBKT has been tested experimentally in the fluid mechanics laboratory. So by comparing the water behavior in this case is the water line trajectory, then the CFKT as a turbine with a new design can be predicted whether it has a better performance than the CBKT or not. In this modeling activity, the focus of the observation is on the movement of water velocity in the blade chamber which will produce momentum or thrust. From the results of this test in general, the Cross Flow Kinetic Turbine has a good performance, because the push of water flow occurs on four blades. Namely the push on two blades on the first stage and push on the two blades on the second stage. Whereas on the Curve Bladed Kinetic Turbine only two blades get a boost. Keep in mind that a push on the blade by the speed of the water will produce momentum that represents the thrust on the turbine blade. From the prototype test results for the 5° runner position the water velocity that produces a boost at CFKT is equal to 3,151 m/s in area a, 4,051 m/s in area f, 2,701 m/s in area b and 4,051 m/s in area e. Whereas for the CBKT the water flow velocity is 2.233 in the area a and the water flow velocity in area b is equal to 2.233 m/s. From this result, it can be seen that the CFKT has a better performance than the CBKT. Overall at each runner's position, the momentum generated at the CFKT is greater than the momentum generated at the CBKT.

Keyword: Kinetic turbine, cross flow, low efficiency, thrust, dynamic computer fluid.

INTRODUCTION

As is known that energy generation is currently being promoted from renewable energy sources. Already many simple turbines were developed just to get electrical energy with low generation efficiency. Many microhydro researchers argue that every increase in efficiency is very expensive. The need for this small generator developed in Indonesia is because the need for electric energy in the countryside is very urgent, both to improve the quality of education and to increase rural productivity. There is an opinion that if electrical energy enters the countryside, the village community will be able to improve their quality of life. [1].

Research on the turbine as the initial driver (Prime movers) is very minimal, because this research is considered less attractive, because of its very low efficiency. As mentioned above, rural communities need to get support to improve their quality of life. Some of the reasons that small and simple power plants still need to be developed, especially for improving the quality of rural life, are because the electricity prices are cheap, construction of turbines and devices is simple, the energy produced can be electrical or mechanical energy which can directly drive mechanical equipment, such as a coffee grinder, rice threshing machine or accumulator filler equipment. Another advantage in operating this small energy generator is the use of free water flow as an energy generator.

Especially in Indonesia, there are still many remote areas that have not yet enjoyed electricity. Much research has been done to improve turbine efficiency. [2, 3, 4]. As mentioned above, the study of small hydroelectric power plants is considered not useful. Though in fact, this kinetic turbine is still very suitable for remote areas. Not too many requirements are needed, head or waterfall height needed does not need to be too high. With the development of technology, humans naturally will increase electricity consumption. [5].

The research that will be carried out is a kinetic turbine that adopts the cross flow turbine. Research on cross flow turbines has been studied with various variations [6, 7]. From one evaluation of the study on the Sutami dam [1], it was observed that water utilization was used to build turbine plants in the irrigation system area included in the Sutami reservoir system (the area after Sutami).

In another reservoir, called a Bening reservoir [8], which regulates irrigation water and has the potential to have an excessive water discharge, it is also evaluated for the possibility of a small power plant installation. This kinetic turbine study was also developed by several researchers [9], and further investigated by several

VOL. 15, NO. 22, NOVEMBER 2020 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

researchers and examined from various perspectives [10, 11, 12].

In general, it is known that this turbine kinetic has a low performance; therefore, some researchers are interested in increasing its efficiency. There have been many attempts to improve turbine efficiency, but it is not clearly explained how to overcome the problem. There are several kinetic turbine studies in terms of the number of turbine blades, the shape of the curved blades, the bowl blades and the effect of the steering angle carried out by Monintja [13] and the hinged blade system carried out by Lempoy [14]. Kinetic turbines are also optimized with the RSM system observed by Boedi [15] and there are some further studies on efforts to increase this efficiency.

A kinetic turbine is a turbine that relies on the river water flow rate or kinetic energy. There are two types of kinetic turbines that have been studied, namely the curve bladed kinetic turbine and the bowl bladed kinetic turbine. Both types of turbines are developed because they are easy to make and easy to maintain and are often found in rural areas [15]. Because of its simplicity, this turbine has a low efficiency. Therefore, through this investigation, it is expected that turbine performance will increase. Investigation of turbine performance in this study was to conduct a simulation investigation using Computer Fluid Dynamic (CFD) technology. The type of kinetic turbine that will be observed is the cross flow kinetic turbine. Cross flow kinetic turbines were chosen in this study because it is a development of a curved blade kinetic turbine that has been studied and the results are quite satisfactory [16]. This investigation was carried out by simulating using CFD with a reason that the research becomes cheaper. Also, that the turbine construction modifications could be done any time, according to what the researcher want to do. Compared with research in the laboratory, the implementation will be very different. The modification process will require costs and the modification process will also be more complicated. Especially if it is connected with the research duration time. The laboratory research will require its own time and energy. The purpose of this study is to simulate the performance of a cross flow kinetic turbine. In order for the simulation results to be valid, the simulation results of the cross flow turbine in the study (Figure-1) will be compared with the conventional curve bladed kinetic turbine. This turbine performance comparison is necessary because the curve bladed kinetic turbine investigation has been carried out in a previous study [16]. Based on the working principle of the cross flow turbine, the cross flow kinetic turbine will do two stages turbine blade propelling. This twice boost is expected to increase the kinetic momentum of this turbine as a whole.

From the total potential of water energy in Indonesia, large-scale hydroelectric power plants that have been utilized are around 3,783 MW, while small-scale power plants are only around 220 MW. The smallest hydroelectric power plant is called the Micro Hydro Power Plant (PLTMH). Where the MHP is a small-scale power plant that utilizes low-speed water energy [15]. Generally small-scale water energy sources utilize water speeds of around 0.01 to 2.8 m/s.

This energy consumption by the end of 2011 was dominated by the industrial sector as the largest consumer, followed by the household sector and the transportation sector.

During 2000-2014, final energy consumption declined mainly in 2005 and 2006. This was caused by increases in fuel prices, which caused a decrease in industrial productivity and a decrease in final energy consumption in the industrial sector in 2005 and in the transportation sector in 2006. Price increases policy, for domestic fuel drives up inflation. Based on data from Bank Indonesia, inflation in January 2005 reached 7.32% and rose to 17.1% in December 2005. [17].

In this study, the performance of cross flow kinetic turbines will be compared to turbine kinetic curves. These two types of turbines will be compared with a simulation process. The comparison of this simulation is done because the curve bladed turbine kinetic laboratory test has been carried out and an experimental performance is obtained. So that by comparing the simulation result, it can be seen, whether a cross flow turbine kinetic has a better performance or not, compared to a curve bladed kinetic turbine.

MATERIAL AND METHODS

As mentioned above, research will be conducted by simulating with a CFD software. In actual conditions, in order to achieve a stable condition, it will always take time to reach a stable condition from an unstable condition. Changes to achieve a stable condition are called transient states.

In preparing this simulation with the CFD software, first what to do is produce the kinetic turbine geometry. The main size of the cross flow kinetic turbine is adjusted to the basic size of the curve bladed kinetic turbine, which has been tested previously in the laboratory and also tested with the CFD simulation. The geometry dimension is seen in Figure-1.

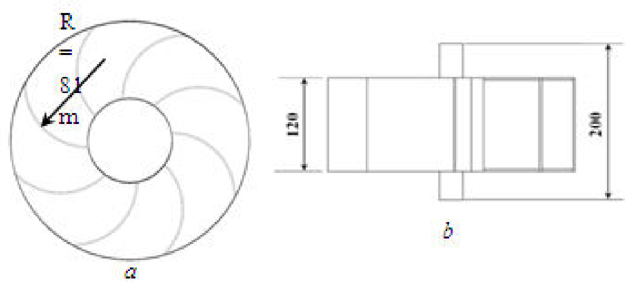


Figure-1. Cross flow kinetic turbine: A -top view; b - side view.

After producing the turbine geometry, the next step is creating the channel geometry as shown in Figure-2. The geometric dimensions of the cross flow kinetic turbine installation are the same as those used in the Curve Bladed Kinetic Turbine (CBKT) CFD simulation. This basic size is made all the same, so that the Cross Flow

ARPN Journal of Engineering and Applied Sciences

Kinetic Turbine (CFKT) test result can be compared with the CBKT test results. The water channel length is about 1500 mm, about 120 mm channel height, a 350 mm channel width and a 14, 5∟ guide blade angle.

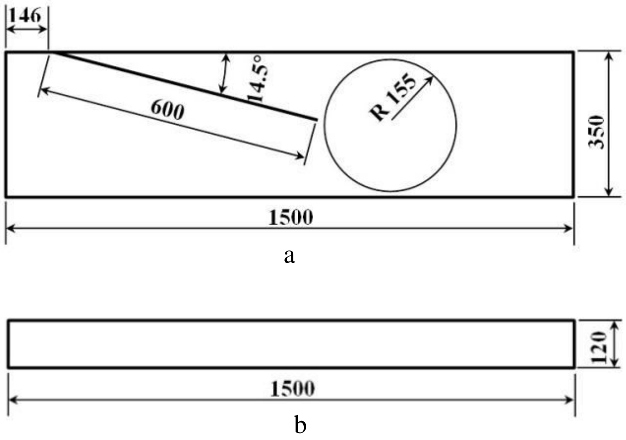


Figure-2. Water flow channel: a -Top view; b - Side View.

It needs to be stressed again that the water channel intended is the channel for the water flow which will be implemented in the simulation as kinetic energy to drive a hydrokinetic turbine.

Next, is to assemble a complete simulation unit. Namely inserting or combining the turbine into the water flow channel. As shown in Figure-3.

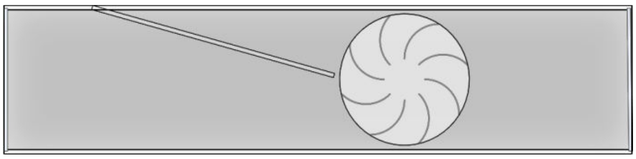


Figure-3. Complete installation (top view).

The next process is the meshing process. The meshing system in this simulation is an automatic meshing system. This automatic meshing system can produce an optimal result; the mesh value is not too firm and not too tight. This mesh system will choose the best total cell number. In this simulation the selected mesh system is 25,700 cells as seen in Figure-4.

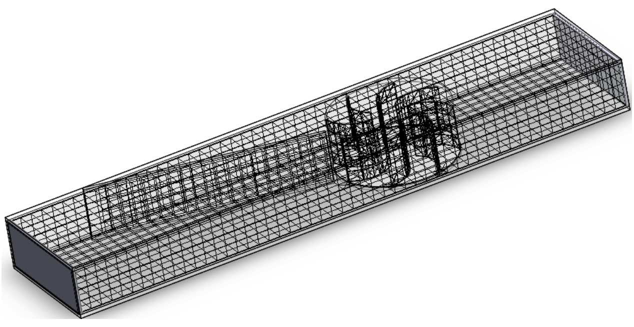
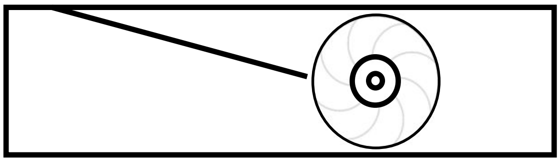


Figure-4. Meshing.

After the meshing value is selected, the next step is to determine the Boundary Condition. In determining this boundary condition two parameters are selected. First is the incoming water flow and the second is the water flow outlet parameters. For the inlet parameter, several conditions can be chosen, such as the inlet mass flow, inlet volume flow or inlet water velocity. Before selecting the channel parameters, the inlet flow field must be determined. After determining the field of inlet flow, then select the inlet flow parameters. In this case the specified inlet chosen is the water flow rate with a value of 0.05 m3/s. For outlet parameters, an environmental pressure of 101325 Pa was chosen at a temperature of 298.2 K (Figure-5).

Inlet Flow rate 0.01 m^3/s



En vir on Pr es s

Figure-5. Hydrokinetic turbine boundary conditions.

The final step is running the active project in the simulation process. After the execution is completed, then determine the trajectory result and determine the surface pressure plot.

Kinetic Turbine

A kinetic turbine is a turbine that work by applying the water flow velocity. The kinetic turbine does not require a high water head. Kinetic turbines are very suitable for river flows in flat areas, especially rural areas. Until now this type of kinetic turbine was called as a water wheel. A water wheel is a very simple kinetic turbine. Waterwheels are still commonly found in Indonesia, used for directly driving simple equipment or as a generator for vehicle battery chargers.

The kinetic turbine works, where direct current flow pushes the turbine blade directly without the speed energy change device. Energy is given to the blade as kinetic energy or speed energy. In vertical kinetic turbines, the water masses directly into the turbine blade. Of course the success of this turbine to spin depends on the turbine blade shape, the turbine blade number and the blade on the runner construction. If the shape and number of blades are inadequate, then the turbine rotation will be blocked and even stop spinning. Therefore, based on this working principle and based on the theory of velocity triangles, the most appropriate blade shape will be obtained. The performance of a water turbine depends on the water flow conditions (water speed and water discharge) and blade angle [18].

There are various types of kinetic turbines, according to the developments carried out for the benefit of an area of use. In this study the kinetic turbine observed was a new design vertical axis turbine kinetic that adopting a cross flow turbine design.

VOL. 15, NO. 22, NOVEMBER 2020 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

Kinetic Turbine Power

The kinetic turbine power produced is as follows:

$$E_a = ½ \cdot \dot{m} \cdot v^2 \quad (1)$$

where:

E_a = Water Energy (joule)
$\dot{m}$ = Water Mass (kg/s)
v = Water flow velocity (m/s)

Water flow power in a specific cross section is as follows:

$$P_a = \frac{1}{2} \cdot \rho \cdot A \cdot v^3 \quad (2)$$

Where:

P_a = Water power (watts)
ρ = Water specific gravity (kg/m^3)

The kinetic energy turbine power generated is as follows:

$$P_t = T \cdot \omega \quad (3)$$

Where:

$$\omega = \frac{2 . \pi . n}{60} \quad (4)$$

where:

Pt = Turbine power (watt)
T = Torque(Nm)
l = Arm length (m)
n = Turbine rotation(rpm)
F = Force(N)

Turbine Efficiency

The kinetic turbine efficiency is the ratio between the incoming water power toward the power generated by the kinetic turbine, as shown in Eq. (5).

$$\eta = \frac{BHP}{WHP} \quad (5)$$

In this case the hydro turbine efficiency is the efficiency with which the hydro turbine converts the water mechanical power into electrical power. This value is used to calculate the nominal hydro power and the actual hydro turbine output in each time step.

Force and Momentum

The force generated by a fluid velocity pushing the turbine blade and will produce a momentum with a magnitude determined as follows:

$$M = m . v \quad (6)$$

Where:

$$m = \rho . Q \quad (7)$$

Then:

$$M = \rho . \ Q . v \quad (8)$$

In accordance with Newton's statements of law, force magnitude is the fluid mass multiplied by fluid acceleration as follows:

$$dF = dm . a = \rho . v . dA . dt \left(\frac{dv}{dt}\right) = \rho . dA . dV \quad (9)$$

Power available in a water stream:

$$P = \eta . \rho . g . h . \dot{q} \quad (10)$$

P = power (watts)
η = turbine efficiency
ρ = water density (kg/m^3)
g = acceleration of gravity (9.81 m/s^2)
$\dot{q}$ = flow rate (m^3/s)
h = head (m).

The Computer Fluid Dynamic Modelling software in this study is used to review the water flow behavior that occurs during each turbine runner movement, because the water flow pushes the blade that produces momentum in each blade. In this model, the water flow formed in the blade and the water pressure that occurs in the runner will be modeled for each 5°runner rotation movement. The number of turbine blades in this study is eight with the same distance from one to another, so that one blade with another blade is separated by 45°. So the total modeling is 45° divided by 5° which is the same as the nine runner angle position modeling.

RESULT AND DISCUSSIONS

The discussion in this section is to compare the movement of water between blades in the Curved Bladed Kinetic Turbine (CBKT) with the movement of water in the Cross Flow Kinetic Turbine (CFKT).

Before the explanation in the discussion is carried out, it is necessary to explain the meaning of the main part of the turbine, to facilitate understanding when the turbine section is explained.

For the cross flow kinetic turbine (CFKT) seen in Figure-6, section 1 is the area of water flow rate input (known as the 1st stage area). Section 2 is the crossing area, where water comes out of the 1st stage and will enter the water input to section 3. (Section 3 is called as the turbine 2nd stage). The symbols a - f are the areas between the blades on a CFKT (Figure-7) and on a CBKT (Figure-8).

ARPN Journal of Engineering and Applied Sciences

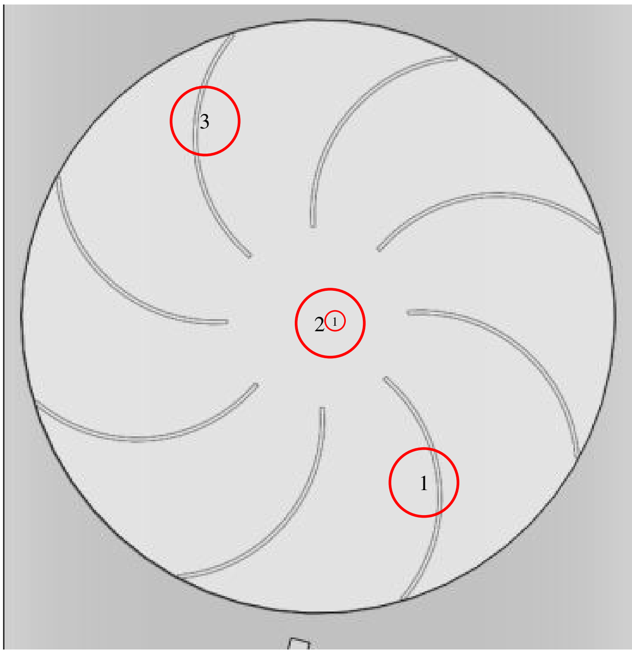


Figure-6. Regions in a CFKT.

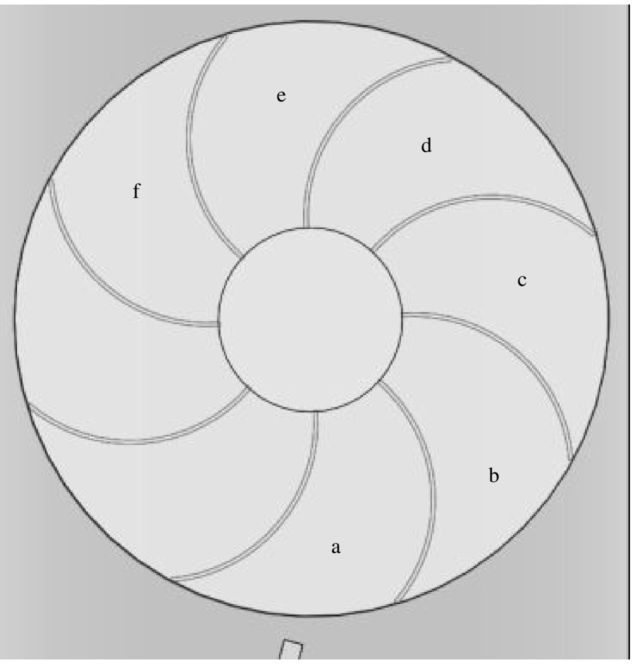


Figure-7. Blade areas in a CFKT.

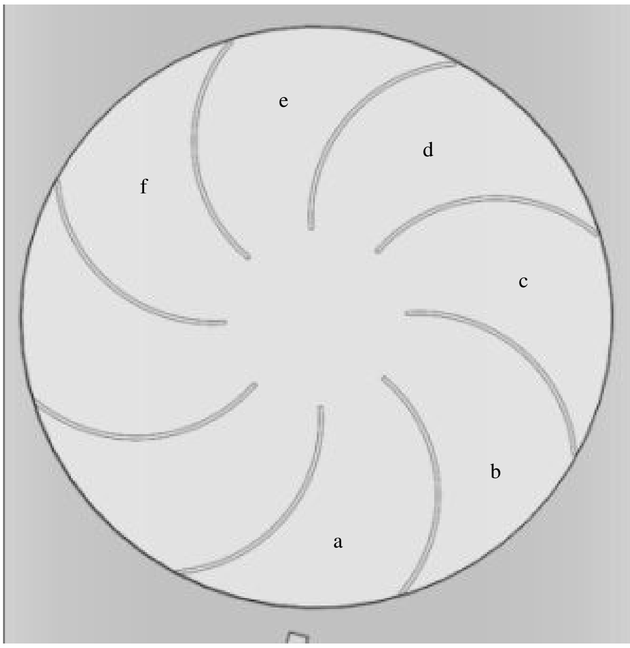


Figure-8. Blade areas in a CBKT

Pressure comparison between the turbine blades and the water trajectory comparison between a CFKT and a CBKT on a runner position of $\alpha = 5°$.

The water trajectory result that occurs in the CFKT at a 5°runner position, could be seen in Figure-9.

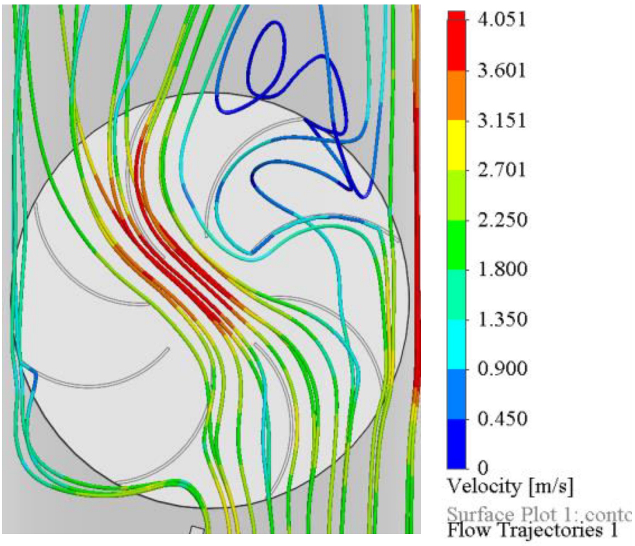


Figure-9. CFKT trajectory on a 5° runner position.

For the CFKT, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 3,151 m/s. While the fluid flow velocity leaving area a is about 4.051 m/s. The fluid velocity entering area f is about 4,051 m/s while the fluid velocity leaving area f is about 4,051 m/s. The fluid velocity entering area b is about 2.701 m/s, and leaving area b with a speed of 3.151 m/s. Fluid velocity entering

ARPN Journal of Engineering and Applied Sciences

area e is about 4,051 m/s while the fluid velocity leaving area e is about 4,051 m/s.

The water trajectory occurs in the CBKT at a 5°runner position result, could be seen in Figure-10.

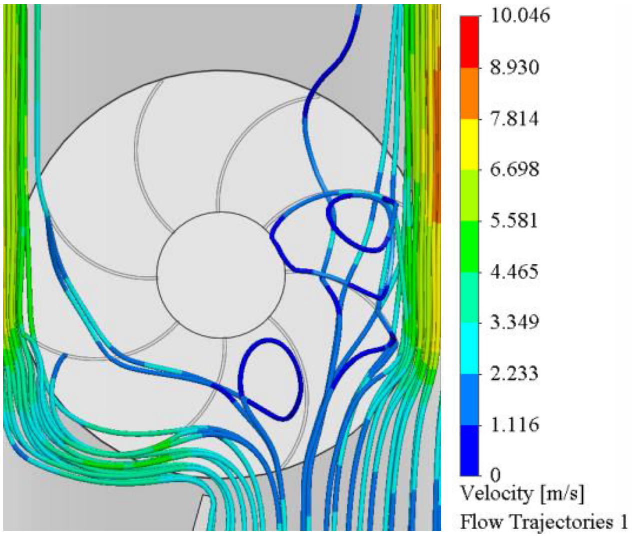


Figure-10. CBKT trajectory on a 5^o runner position.

The flow trajectory line is not clear enough (Figure-10), the maximum water speed in the area a = 2.233 m/s and around 2.233 m/s in the area b.

For the water trajectory occurs in the CFKT at a 10∟runner position the result could be seen in Figure-11.

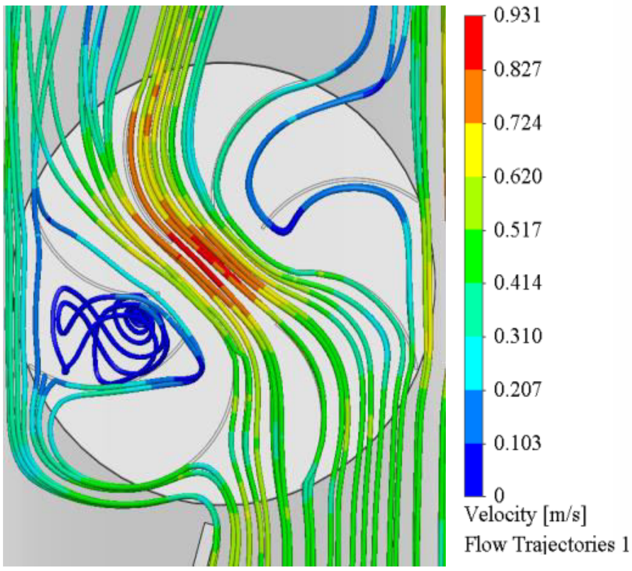


Figure-11. CFKT trajectory on a 10^o runner position.

For the CFKT at a 10∟ runner position seen in Figure-11, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 3,601 m/s. While the fluid flow velocity leaving area a is about 4.051 m/s. The fluid velocity entering area f is about 4,051 m/s while the fluid velocity leaving area f is about 4,051 m/s. The fluid velocity entering area b is about 2.701 m/s, and leaving area b with a speed of 3.151 m/s. Fluid velocity entering area e is about 4,051 m/s while the fluid velocity leaving area e is about 4,051 m/s.

For the water trajectory occurs in the CBKT at a 10∟runner position, the result could be seen in Figure-12.

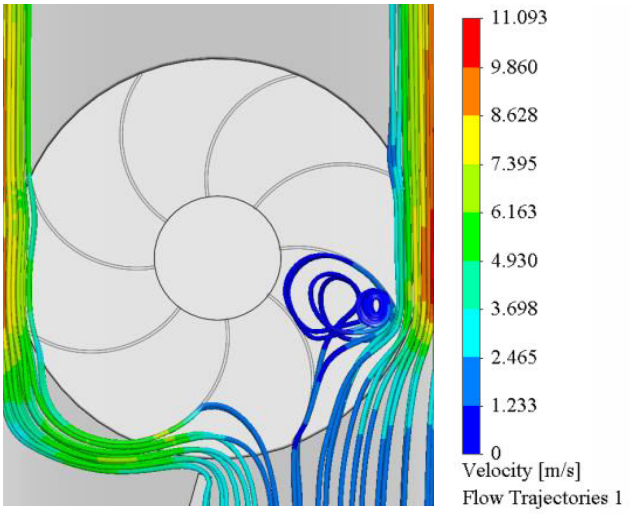


Figure-12. CBKT trajectory on a 10^o runner position.

The flow trajectory line is not clear enough (Figure-12). The maximum water speed in section a = 2,465 m/s and in area b is around 2,465 m/s.

The water trajectory occurs in the CFKT at a 15∟runner position result, could be seen in Figure-13.

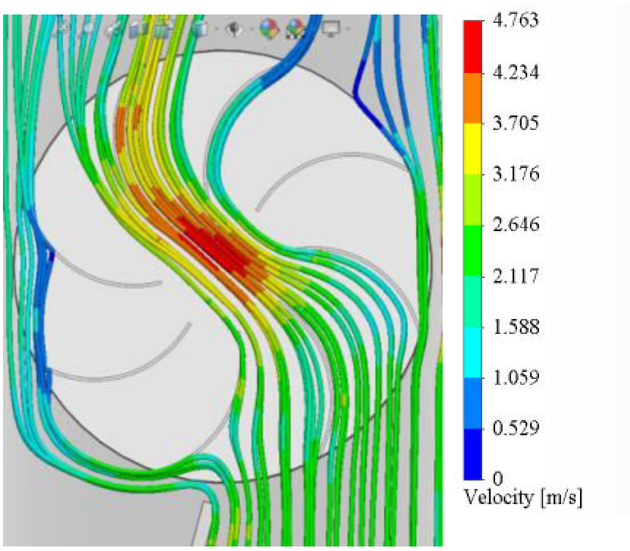


Figure-13. CFKT trajectory on a 15^o runner position.

For the CFKT in Figure-13, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 3.176 m/s. While the fluid flow velocity leaving area a is about 3.705 m/s. The fluid velocity entering area f is about 3.705 m/s while the fluid velocity leaving area f is about 3.705 m/s. The fluid velocity entering area b is about 3.176 m/s, and leaving area b with a speed of 3.705 m/s. Fluid velocity entering

ARPN Journal of Engineering and Applied Sciences

area e is about 4.705 m/s while the fluid velocity leaving area e is about 3.176 m/s.

The water trajectory that occurs in the CBKT at a 15∟runner position result, could be seen in Figure-14.

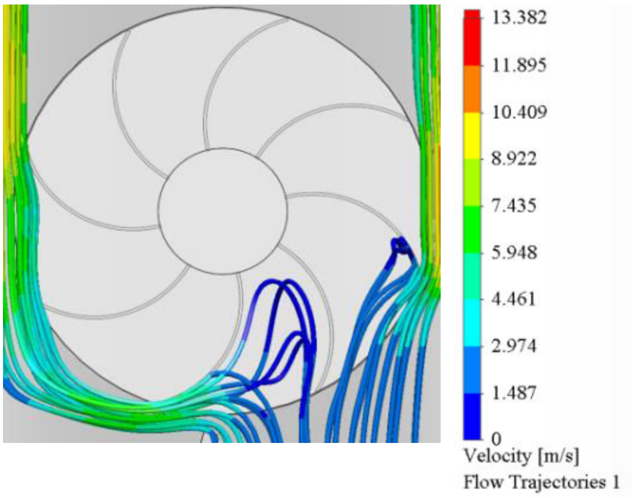


Figure-14. CBKT trajectory on a 15° runner position.

The flow trajectory line is not clear (Figure-14). The maximum water speed in section a = 2,974 m/s and the maximum water speed in area b is around 4,461 m/s.

The water trajectory occurs in the CFKT at a 20∟runner position result, could be seen in Figure-15.

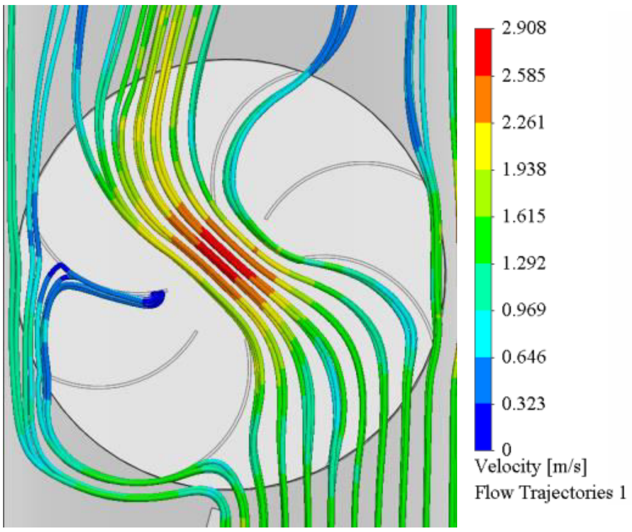


Figure-15. CFKT trajectory on a 20° runner position.

For the CFKT in Figure-15, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 1.938 m/s. While the fluid flow velocity leaving area a is about 2.261 m/s. The fluid velocity entering area f is about 2.261 m/s while the fluid velocity leaving area f is about 1.938 m/s. The fluid velocity entering area b is about 1.615 m/s, and leaving area b with a speed of 1.938 m/s. Fluid velocity entering area e is about 4.705 m/s while the fluid velocity leaving area e is about 3.176 m/s.

The water trajectory occurs in the CBKT at a 20∟runner position result, could be seen in Figure-16.

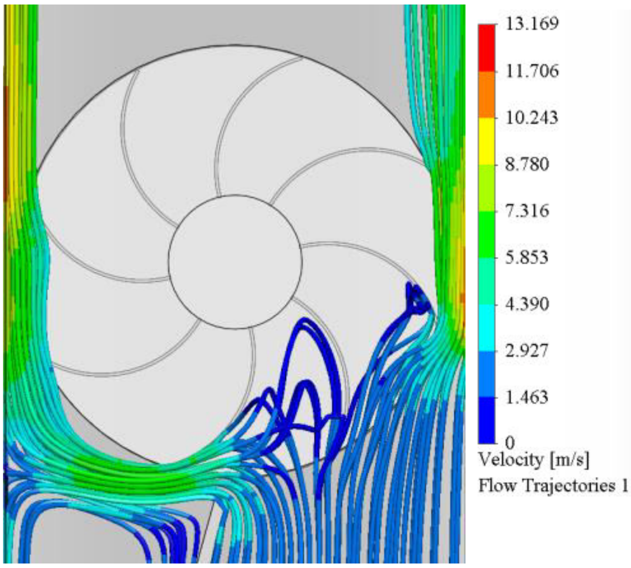


Figure-16. CBKT trajectory on a 20° runner position.

The flow trajectory line is not clear (Figure-16). The maximum water speed in the area a = 4.390 m/s and the maximum water speed in the area b is around 2.927 m/s.

For the water trajectory occurs in the CFKT at a 25° runner position result, could be seen in Figure-17.

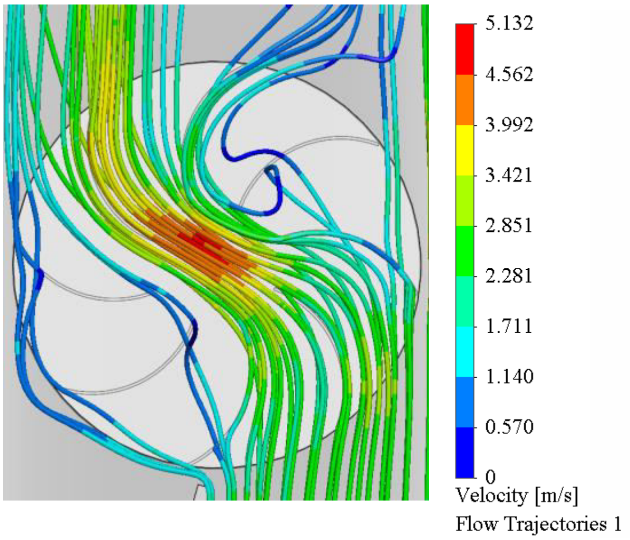


Figure-17. CFKT trajectory on a 25° runner position.

For the CFKT in Figure-17, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 2.281 m/s. While the fluid flow velocity leaving area a is about 3.992 m/s. The fluid velocity entering area f is about 3.992 m/s while the fluid velocity leaving area f is about 3.992 m/s. The fluid velocity entering area b is about 3.421 m/s, and leaving area b with a speed of 3.992 m/s. Fluid velocity entering area e is about 3.992 m/s while the fluid velocity leaving area e is about 2.281 m/s.

For the water trajectory occurs in the CBKT at a runner 25°position, the result could be seen in Figure-18.

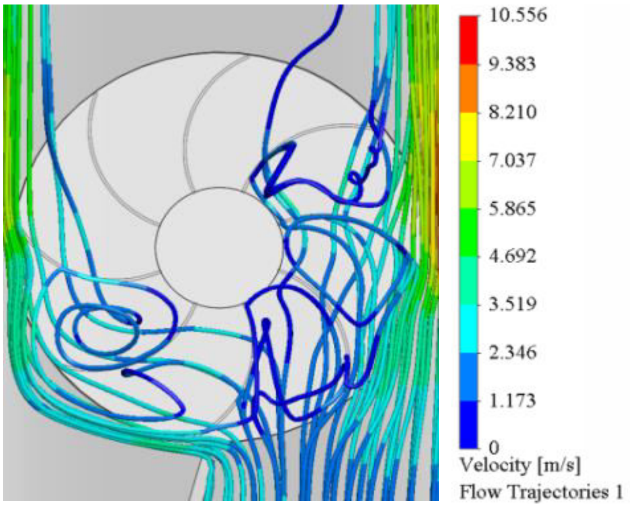


Figure-18. CBKT trajectory on a 25° runner position.

The flow trajectory line is not clear enough (Figure-18). The maximum water speed in the area a = 2,346 m/s in the maximum water speed in the area b is around 3,519 m/s.

For the water trajectory occurs in the CFKT at a 30°runner position. The result could be seen in Figure-19.

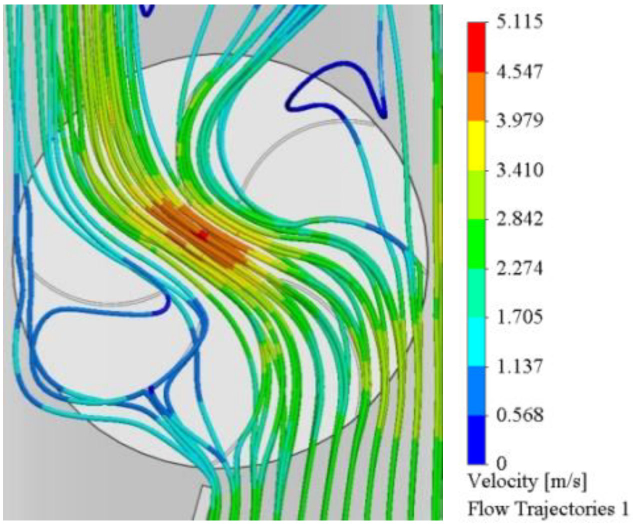


Figure-19. CFKT trajectory on a 30° runner position.

For the CFKT in Figure-19, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 3.410 m/s. While the fluid flow velocity leaving area a is about 3.979 m/s. The fluid velocity entering area f is about 3.979 m/s while the fluid velocity leaving area f is about 3.410 m/s. The fluid velocity entering area b is about 3.410 m/s, and leaving area b with a speed of 3.979 m/s. Fluid velocity entering area e is about 3.979 m/s while the fluid velocity leaving area e is about 3.979 m/s.

For the water trajectory occurs in the CBKT at a 30° runner position, the result could be seen in Figure-20.

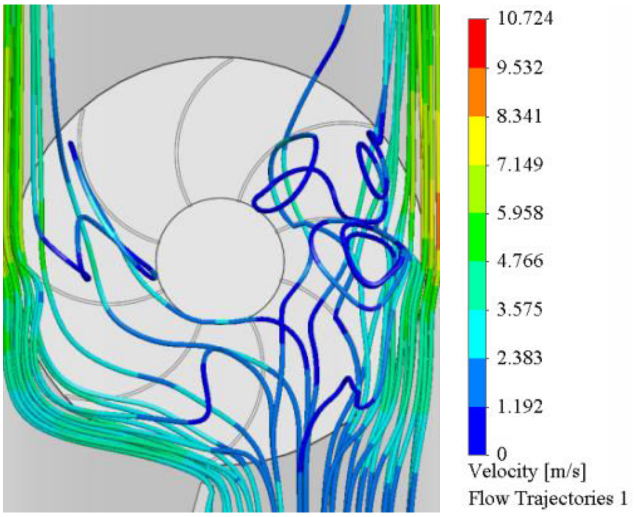


Figure-20. CBKT trajectory on a 30° runner position.

The flow trajectory line is not too clear (Figure-20), the maximum water speed in the area a = 2,383 m/s and in the area b is around 3,575 m/s.

The water trajectory occurs in the CFKT at a 35°runner position result, could be seen in Figure-21.

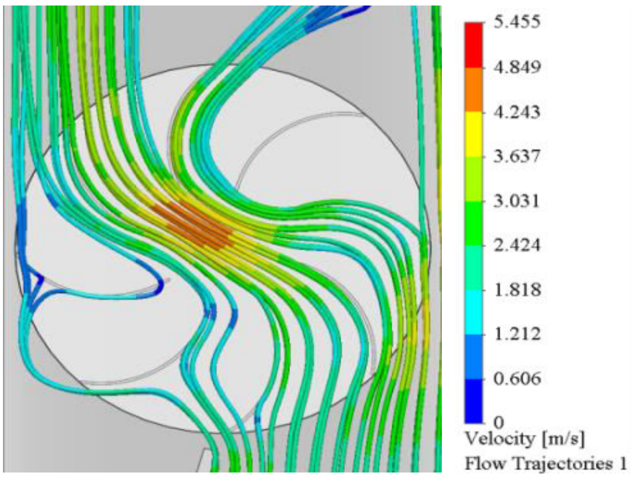


Figure-21. CFKT trajectory on a 35° runner position.

For the CFKT in Figure-21, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 3.031 m/s. While the fluid flow velocity leaving area a is about 4.243 m/s. The fluid velocity entering area f is about 4.243 m/s while the fluid velocity leaving area f is about 4.243 m/s. The fluid velocity entering area b is about 3.0310 m/s, and leaving area b with a speed of 4.243 m/s. Fluid velocity entering area e is about 4.243 m/s while the fluid velocity leaving area e is about 3.637 m/s.

For the water trajectory occurs in the CBKT at a 35°runner position result, could be seen in Figure-22.

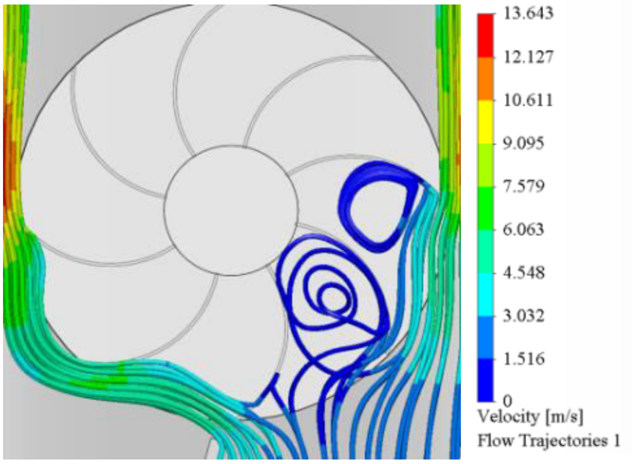


Figure-22. CBKT trajectory on a 35° runner position.

The flow trajectory line is not clear (Figure-22), the maximum water speed in the area a = 1.516 m/s and in the area b is around 3.032 m/s.

For the water trajectory occurs in the CFKT at a 40°runner position result, could be seen in Figure-23.

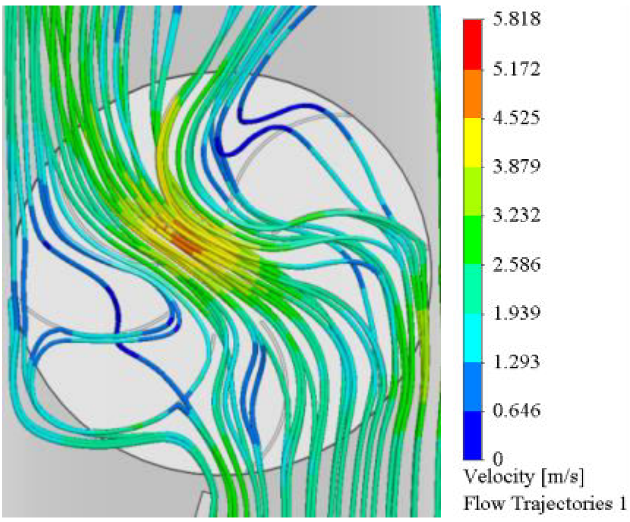


Figure-23. CFKT trajectory on a 40° runner position.

For the CFKT in Figure-23, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 3.232 m/s. While the fluid flow velocity leaving area a is about 3.879 m/s. The fluid velocity entering area f is about 3.879 m/s while the fluid velocity leaving area f is about 3.232 m/s. The fluid velocity entering area b is about 2.586 m/s, and leaving area b with a speed of 4.525 m/s. Fluid velocity entering area e is about 4.525 m/s while the fluid velocity leaving area e is about 3.879 m/s.

The water trajectory occurs in the CBKT at a 40°runner position result, could be seen in Figure-24.

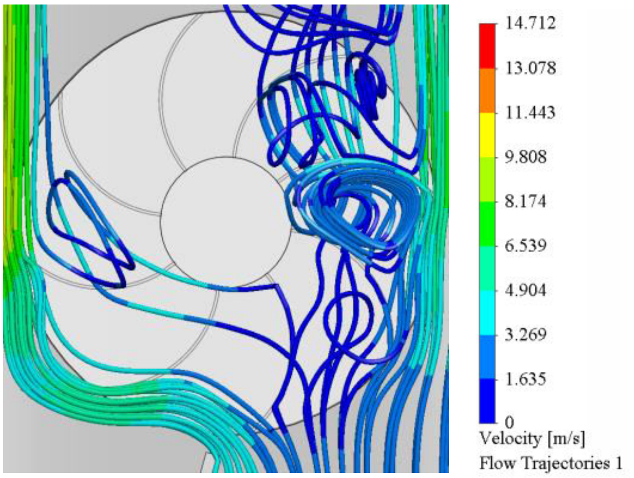


Figure-24. CBKT trajectory on a 40° runner position.

The flow trajectory line is not clear (Figure-24), the maximum water speed in the area a = 1,635 m/s and in the area b is around 3.269 m/ s.

The water trajectory that occurs in the CFKT at a 45°runner position result could be seen in Figure-25.

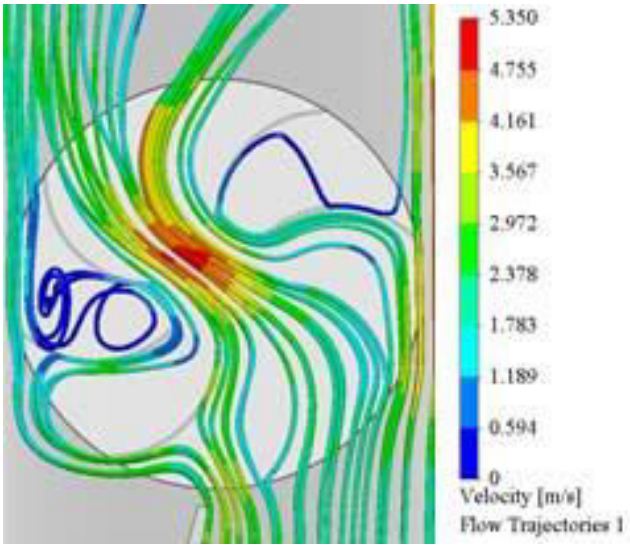


Figure-25. CFKT trajectory on a 45° runner position.

For the CFKT in Figure-25, there are two blades that get a boost from the water speed on the inlet section 1 with a water velocity of around 2.378 m/s. While the fluid flow velocity leaving area a is about 3.567 m/s. The fluid velocity entering area f is about 3.567 m/s while the fluid velocity leaving area f is about 3.755 m/s. The fluid velocity entering area b is about 3.567 m/s, and leaving area b with a speed of 4.161 m/s. Fluid velocity entering area e is about 4.525 m/s while the fluid velocity leaving area e is about 4.755 m/s.

The water trajectory that occurs in the CBKT at a 45°runner position result could be seen in Figure-26.

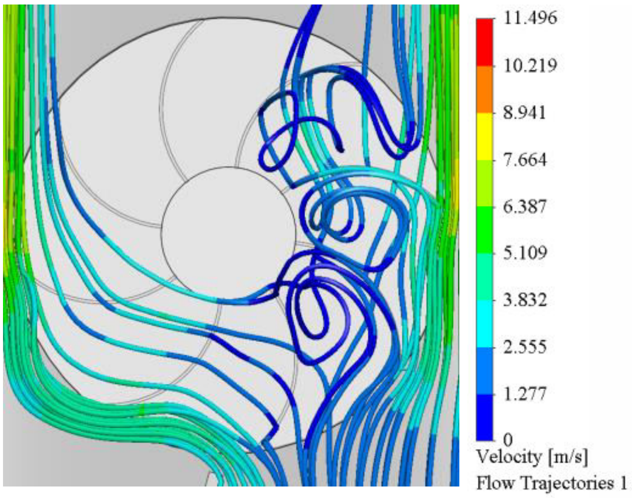


Figure-26. CBKT trajectory on a 45° runner position.

The flow trajectory line could be seen in Figure-26. The maximum water speed in the area a = 2,555 m/s and in the area b is around 2,555 m/s.

CONCLUSIONS

This research was conducted by testing the cross flow kinetic turbine by means of simulation using CFD software. Simulation results for the CFKT will be compared with the CBKT simulation results. The reason for comparing with the CBKT simulation results, is because this CBKT has been tested experimentally on a laboratory scale and verified virtually with CFD software. So by comparing the results of this CFD simulation, it can be seen whether the CFKT has a better performance than the CBKT performance. It is estimated that the CFKT will improve the kinetic turbine performance because water speed will generate a momentum for the two stages of the turbine blade impulse. Generating momentum on two turbine blade boosts except increasing the turbine efficiency will also result in a more stable turbine rotation. In this model, the focus of the observation is on the water trajectory that occurs in the turbine blade. This water speed will provide a momentum and will generate thrust on the turbine runner. From the observations, for each movement of 5° runner position on the CFKT, it appears that there are four turbine blades that will gain momentum from the speed of the water. Whereas in the CBKT, the push of water velocity only occurs on one or two turbine blades. This condition is thought to be one of the causes of the low efficiency of the CBKT turbine. By looking at the whirlpool in the runner turbine, in this case the movement of the flow of water that occurs in the blades, then there appears a big push. While on the other hand, produces a small push. Furthermore, by looking at the behavior of the water trajectory, there is a large amount of water flow rate, would drastically lower the pressure because it does not enter the turbine blade area but instead directly switches to the channel output. This is thought to be one of the causes of the CBKT low efficiency. Whereas at the CFKT, water velocity does not immediately leave the turbine area, but continues to push on the second stage. In conclusion, from the prototype test results for the 5° runner position the water velocity that produces a boost at the CFKT is equal to 3.151 m/s in area a, 4,051 m/s in area f, 2,701 m/s in area b and 4,051 m/s in the area e. Whereas for the CBKT the water flow velocity is 2,233 m/s in the area a and the water flow velocity in area b is equal to 2,233 m/s. From this result, it can be seen that the CFKT has a better performance than the CBKT. Overall at each runner's position, the momentum generated at the CFKT is greater than the momentum generated at the CBKT. So, in general, it could be concluded that the CFKT has a better performance compared to the CBKT. This CFKT performance result always occurs at every runner position angle.

REFERENCES

[1] Rispiningtati Soenoko R. 2015. Regulation of Sutami Reservoir to Havea Maximal Electrical Energy. International Journal of Applied Engineering Research 10(12): 31641-31648. (6)

[2] Soenoko R. 2015. Design optimization to increase across flow turbine performance: A review. International Journal of Applied Engineering Research. 10(18): 38885-38890. (9)

[3] Soenoko R. 2016. First stage cross flow turbine performance, International Journal of Applied Engineering Research. 11(2): 938-943. (6)

[4] Soenoko R, Purnami, Fransisca Gayuh Utami Dewi. 2017. Second stage Cross Flow Turbine Performance., ARPN Journal of Engineering and Applied Sciences. 12(6): 1772-1779.

[5] Kautsari Anggun Karisma, Ghozali Maski, Iswan Noor. 2016. Analysis of Electricity Consumption Behaviour: Case Study of Non Business and Business Household in Malang City. International Journal of Social and Local Economic Governance (IJLEG) E-ISSN: 2477-1929; 2(2): 168-176.

[6] Haurissa J., Wahyudi S., Irawan Y.S., Soenoko R. 2012. The Cross Flow Turbine Behavior towards the Turbine Rotation Quality, Efficiency, and Generated Power. Journal of Applied Sciences Research. 8(1): 448-453.

[7] Choi Y.D., Son S.W. 2012. Shape Effect of Inlet Nozzle and Draft Tube on the Performance and Internal Flow of Cross-Flow Hydro Turbine, Journal of the Korean Society of Marine Engineering. 36(3): 351-357.

[8] Rispiningtati Soenoko R. 2015. Optimization Operationof Bening Reservoirto Maximize Irrigation Allocation. International Journal of Applied Engineering Research. 10(13): 33197-33201. (5)

[9] Brian Kirke. 2003. Developments in Ducted Water Current Turbines (This paper was originally written in 2003 and published on www.cyberiad.net. This version includes some updates on tests conducted in 2005) Sustainable Energy Centre, University of South Australia, Mawson Lakes, SA 5095, Australia. Tidal paper 25-04-06

[10] Lempoy K.A., Soenoko R., Wahyudi S., Choiron M.A. 2015. Bowl Bladed Vertical Shaft Kinetic Turbine Performance Using Response Surface Methodology (RSM). International Journal of Applied Engineering Research 10(7): 16399-16407 (7).

[11] Boedi S.D., Soenoko R., Wahyudi S., Choiron M.A. 2015. An Outer Movable Blade Vertical Shaft Kinetic Turbine Performance. International Journal of Applied Engineering Research. 10(4): 8565-8573 (9).

[12] Monintja N.C.V., Soenoko R., Wahyudi S., Irawan Y.S. 2014. The Influence of Flow Steering Angleonthe Performance of a Cup-bladed Kinetic Turbine. International Journal of Applied Engineering Research. 9(20): 7481-7489.

[13] Monintja N.C.V., Soenoko R., Wahyudi S., Irawan Y.S. 2014. The Vertical Shaft Kinetic Turbine Optimizing Using Response Surface Methodology. International Journal of Applied Engineering Research. 9(21): 8841-8856. (7).

[14] Lempoy K. A., Soenoko R., Wahyudi S., Choiron M. A. 2017. Response Surface Methodology (RSM) Application toward the Performance of a Vertical Shaft Hinged Arc Blade Kinetic Turbine. Journal of Engineering Science and Technology. 12(8): 2175-2168.

[15] Indonesia Energy Outlook 2017, Agency for the Assessment and Application of Technology; Indonesia.

[16] Soenoko R., Setyarini P. H., Gapsari F. 2018. Eight Curved Bladed Kinetic Water Turbine Performance. ARPN Journal of Engineering and Applied Sciences. 13(6): 2138-2147.

[17] Supriono Al-Musadieq, Topowijono. 2016. Marketing Innovation through the Identification of Business Model Canvas in order to enhance the Competitiveness of Micro, Small and Medium Enterprises in Bondowoso Regency. International Journal of Social and Local Economic Governance (IJLEG). 2(2): 118-126.

[18] Kennie Abraham Lempoy, Rudy Soenoko, Slamet Wahyudi, Moch Agus Choiron. 2019. Movable Blade Vertical Shaft Kinetic Turbine Visual Observation. Eastern-European Journal of Enterprise Technologies. 2(8), (98). DOI: https//doi.org/10.15587/1729-4061.2019.163418