

## Numerical Study the Effect of Blade Number on Archimedes Screw Turbine Utilized in Pico-Hydro in Horizontal Position

Fajar Anggara<sup>1</sup>

<sup>1</sup>Program Studi Teknik Mesin, Fakultas Teknik, Universitas Mercu Buana Jakarta

E-mail: fajar.anggara@mercubuana.ac.id

**Abstrak**--Pemanfaatan Archimedes Screw Turbine (AST) pada pikohidro semakin populer karena mampu menghasilkan torsi besar pada debit yang kecil pada skala piko-hidro dan sangat ramah terhadap organisme sekitar semisal ikan yang mampu melewati AST. Penelitian ini menggunakan CFD untuk mengamati performa AST akibat dari penambahan jumlah sudu turbin. Independency of mesh dilakukan untuk menentukan jumlah mesh yang efisien dan didapat sekitar 80 ribu mesh. Variasi A, B dan C berturut-turut dengan jumlah sudu 4,6 dan 9 ulir sudu digunakan untuk melihat torsi, pressure loss dan kontur kecepatan yang dihasilkan. Didapat dengan meningkatkan jumlah ulir sudu turbin menghasilkan torsi yang paling besar. Hal ini disebabkan karena sudu yang banyak mampu menyerap energi kinetik air yang juga lebih banyak. Meskipun demikian, pressure loss yang dihasilkan pada sudu tersebut pun juga yang paling besar. Hal ini disebabkan karena kesetimbangan energi, dimana delta tekanan yang besar sebanding dengan penyerapan energi oleh turbin.

**Kata kunci:** CFD, AST, Sudu, pikohidro, Archimedes

**Abstract**— Application of Archimedes Screw Turbine (AST) on pico-hydro has been very popular due to its capability to has a high torque with small flow rate and environmentally friendly to biotic of vicinity. This study uses CFD to investigate AST performance when the number of blades is varied. Independency of mesh is conducted to find an efficient number of mesh and it is obtained at 80 thousand mesh. Variation A, B and C respectively has 4,6, and 9 blades varied to analyze the correspondence with torque, pressure loss and velocity contour. The results show that with a higher number of blades would make higher torque. This is because energy extraction from water would be more efficient at higher number blade. However, the side back at such number would arise such as the higher-pressure loss. This is corresponding to energy balance where a high delta pressure is proportional with a generated energy turbine.

**Keywords:** CFD, AST, Blade, Pico-hydro, Archimedes

### 1. INTRODUCTION

The use of energy every year is always increasing due to the growth of the economic sector and population[1]. Several efforts have been made such as saving in the energy sector and optimizing the use of renewable energy in order to meet the burden of energy needs. Some energy saving applications such as Latent Heat Thermal Energy Storage (LHTES)[2] and utilization of other energies to increase efficiency [1], [3].

CFD simulation is a method used to study physical phenomena from fluid dynamics, heat transfer, aerodynamics and important stages for model design development [4], [5].

Utilization of renewable energy has been applied to several fields such as wind turbines, water turbines, biofuels and other fields. The use of pico-hydro scale water turbines is applied to water source areas that have low discharge. The use of Archimedes Screw Turbine (AST) has been widely applied and utilized in pico-hydro hydropower applications.

As has been done by a study by Kadir et al [6]

explains the future challenges of pico-hydro AST in Malaysia. The study states that the obstacles faced are in the form of a lack of water sources, a lack of experts and a lack of regulations that support the use of renewable energy, both incentives and restrictive. Apart from the difficulties faced, Malaysia intends to target 500 MW of energy production and participate in pico-hydro. This indicates that Malaysia has made preparations to implement renewable energy applications.

The study conducted by Siswanto et al,[7] observing the effect of the angle of inclination of the penstock on the leakage overflow phenomenon [7], [8]. The study states that the greater the inclination of angle, the greater the amount of electrical energy in the generator but the turbine efficiency is small. This is because at a large angle it produces a large leakage overflow as well. So that the friction that is generated is getting bigger and becomes a big energy loss too.

To complement the AST performance study, this study focuses on the effect of the number of turbine screw blades on the torque generated by AST on horizontal penstock.

## 2. RESEARCH METHODOLOGY

This section describes the stages carried out in the study. The complete stages can be seen in Figure 1.

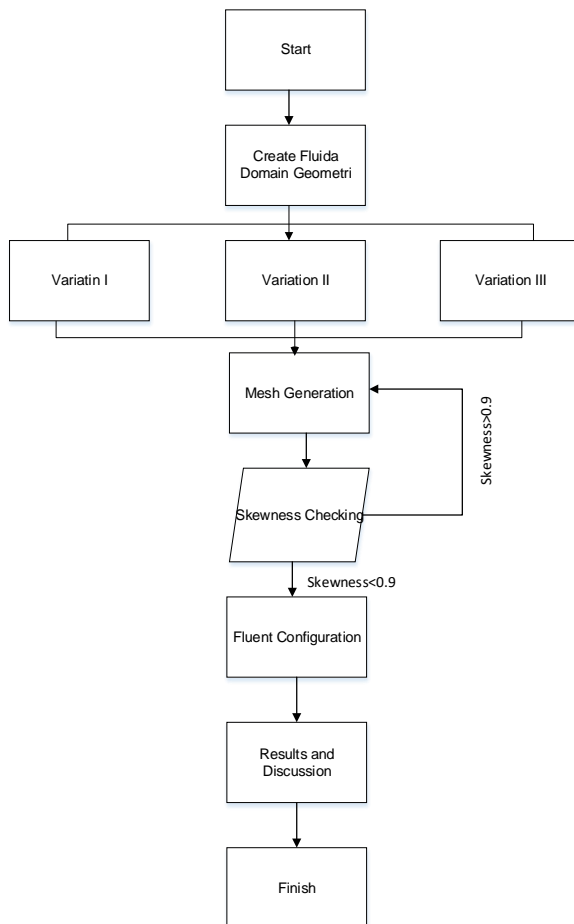


Figure 1. Research Stages

In Figure 1, it can be seen that the initial steps taken are to create a fluid domain in the simulation. After that, it is continued to make geometry according to the variations in the study. Software used to draw geometry using CAD. Then making the mesh is done as well as checking the skewness parameter. If it is more than 0.9 then it will be corrected again if it is less, then proceeding to the simulation preparation stage at Fluent.

Independency of mesh is also carried out to obtain the optimal number of mesh by comparing the number of meshes and the resulting torque.

### 2.1 Geometry

The geometric shape of AST can be seen in Figure 2. In the figure, it can be seen that A is the length of the diameter, B is the distance of the screw blade and C is the length of the AST. Meanwhile, the division of zones and boundary conditions can be seen in Figure 3.

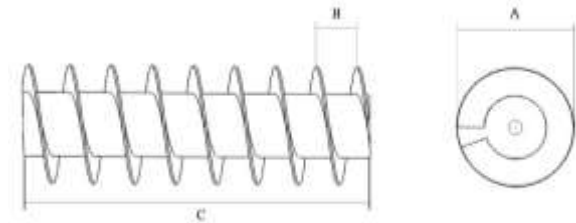


Figure 2. AST Geometry

This simulation uses the concept of multiple references where a zone must be defined and becomes a moving zone. This is because the zone is experiencing movement, especially in this case the rotating zone which is the area of the turbine blade in Figure 3.

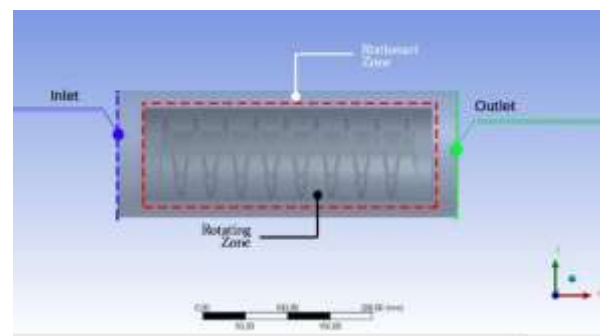


Figure 3. Zone and Boundary Condition

The inlet becomes the inlet in the form of water and the outlet is the place for its output. Because the stationary and rotating zones are separate, an interface is made between them and the interface used is of a match type. The fluid parameters entered using the default water and more details are shown in Table 1.

Table 1. Fluid Parameter

Parameter	Description
Fluid	water
V, Flowrate	0.0033 m <sup>3</sup> /s
Density	998.2 kg/m <sup>3</sup>
Viscosity	0.001003 kg/m.s

### 2.2 Turbine Blade Variation

The variations used in the number of AST blade can be seen in more detail in Table 2.

Table 2. Geometry Variation

Turbine Parameter	I	II	III
A/B	0.3	0.5	0.75
Blade number	9	6	4
A	101.6 mm	101.6 mm	101.6 mm
C	300 mm	300 mm	300 mm

In Table 2, it can be seen that the naming of variations I, II and III represents the variation in the number of threads 9.6 and 4. The geometric size of the length of diameter A and length of AST C are made the same. Variations are

distinguished based on the ratio between A and B (distance between threads). This is intended to get the best ratio between A and B.

The Tip Speed Ratio (TSR) equation is used to determine the angular velocity of the turbine rotation. The equation is shown as follows[9]:

$$TSR = \frac{\omega R}{v} \tag{1}$$

Where  $\omega$  is the angular velocity of the turbine, R is the radius of AST and V is the inlet velocity of the wind. The turbine power can be calculated using Equation (2) [9]:

$$Wt = T \times \omega \tag{2}$$

Where T is the value of torque obtained and  $\omega$  is the angular rotation of the turbine.

### 2.3 Numerical Model and Simulation Configuration

The governing equation used to model the conservation of time and momentum is shown as follows [9]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \tag{3}$$

$$\frac{\partial \rho v}{\partial t} + \nabla \rho v \vec{V} = -\nabla P + \mu \cdot \nabla^2 v + \rho g - S \tag{4}$$

Equation (3) is the conservation of mass equation while Equation (4) is the conservation of momentum. Where  $\rho$  is the density and  $\mu$  is the viscosity of the fluid [9]. The energy conservation equation is not used because it assumes that there is no transfer of heat energy from within or outside the system boundaries. The turbulence model used is the k-e model [10]. The boundary conditions are entered in accordance with Table 1.

The number of meshes in the simulation greatly affects the results of the simulation. If too many meshes are used, the computation load on the computer will be heavier and longer. However, using a small number of mesh will reduce the accuracy of the simulation so that an independence of mesh study is needed. The results of the independence of mesh can be seen in Figure 4.

In Figure 4, you can see how the number of meshes affects the resulting torque. The results show that the number of mesh 89 thousand is not much different from the number of mesh 140 thousand. This means that the addition of the number of meshes from 89 thousand to 140 thousand has not changed. So that in this study, 89 thousand mesh were used.

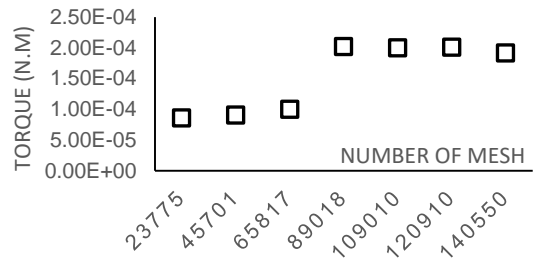


Figure 4. Independency of Mesh

### 3. RESULTS AND DISCUSSION

This section will explain the characteristics of the turbine by drawing a TSR graph with torque and then displaying the speed and pressure contours.

The resulting velocity contour can be seen in Figure 5.

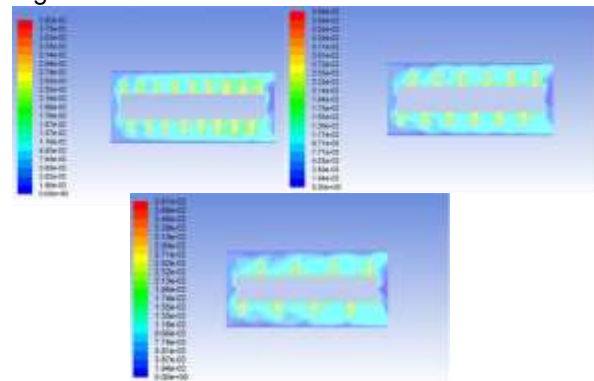


Figure 5. Axial Velocity Contour

It can be seen in variation I that the water velocity is drastically reduced when compared to other variations. This is because a lot of water collides with the turbine blades so that a lot of the kinetic energy of the water is transferred to the turbine blades.

The pressure contour on the axial cross section of the turbine can be seen in Figure 6. It can be seen that the pressure drop is very drastic in variation I. While the other variations do not experience a very large pressure drop. This shows that the greatest flow resistance is in variation I because of the many blades. Due to the many collisions between the water and the blade, the pressure is reduced as a compensation for the transfer of kinetic energy. The results of the pressure drop are shown in Table 3.

AST characteristics can be seen in the graph presented in Figure 7. It can be seen that the relationship between TSR and torque is directly proportional. However, it can be seen that the variation I shows the highest resulting torque. Even at a low TSR, the torsion results show the opposite.

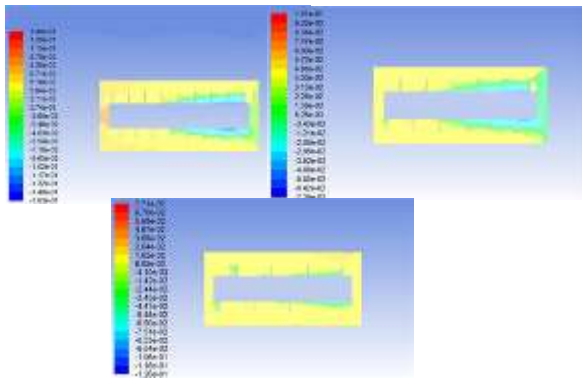


Figure 6. Axial Pressure Contour

study because water fills the entire penstock. It is necessary to do further research where the pressure drop in variation III is greatest. It is necessary to do an experimental test to validate the simulation results.

Table 3. Result of Pressure Drop

Description	I	II	III
Pin	0.097 Pa	0.046 Pa	0.02 Pa
Pout	0.046 Pa	0.041 Pa	0.002 Pa
Delta P	0.049 Pa	0.005 Pa	0.0001 Pa

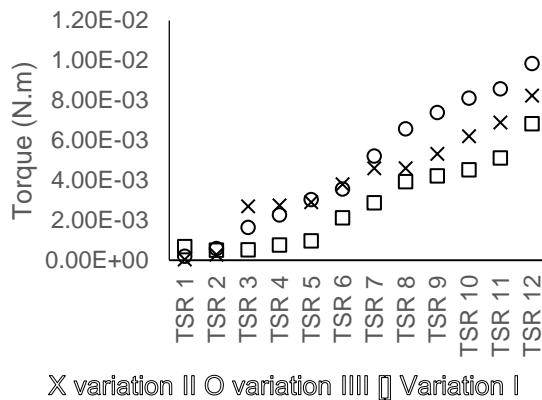


Figure 7. TSR Versus Torque

In TSR 1 and 2, it can be seen that the resulting torque is not much different. This is because the water is still unable to move the turbine. As the mass of incoming water increases, the turbine blades start to rotate and generate torque.

At TSR 3 to TSR 6, variation II produces the greatest torque, while variation III is in the second place. This is because the mass of water entering variation III is still unable to rotate the turbine optimally. Seen in TSR 7 where variation III is able to exceed the torque of variation II.

**4. CONCLUSION**

From the results of this study, the influence of the number of turbine blades with the torque produced. It can be seen that the relationship between the addition of turbine blades is proportional to the torque produced. However, the effect of mass accumulation also determines the turbine rotation. It can be seen that the low TSR variation II is able to outperform variation III in the resulting torque. The phenomenon of leakage overflow is still not clearly seen in this

## REFERENCE

- [1] F. Anggara, "DENGAN ANALISA NUMERIK TERHADAP POLA ALIRAN DAN KECEPATAN," *JTM*, vol. 09, no. 2, pp. 124–128, 2020.
- [2] F. Anggara, R. A. Anugrah, and H. Pranoto, "Thermal energy storage using horizontal shell-tube heat exchanger: Numerical investigation on temperature variation of HTF," *Int. J. Renew. Energy Res.*, vol. 9, no. 4, pp. 2112–2117, 2019.
- [3] F. Anggara, "Numerical Study: Comparison The Effect Of Ratio Length And Diameter Guide Vane Of Air Flow On Outlet Ducting," *SJME Kinemat.*, vol. 5, no. 2, pp. 150–159, 2020, doi: 10.20527/sjmekinematika.v5i2.178.
- [4] R. Amaratu and F. Anggara, "Investigation Of Numerical Analysis In The Effect Of Comparison Of Inlet And Outlet Diameter Of Guide Vane On Velocity Profile," *SJME Kinemat.*, vol. 5, no. 1, pp. 1–10, 2020, doi: 10.20527/sjmekinematika.v5i1.98.
- [5] F. Anggara, "NUMERICAL STUDY OF COMPARISON THE EFFECT OF FRICTION AND RESTITUTION COEFFICIENT ON SEPARATION," *SINTEK J.*, vol. 14, no. 2, pp. 80–84, 2020, doi: 10.24853/sintek.14.2.80-84.
- [6] A. Kadier, M. S. Kalil, M. Pudukudy, H. A. Hasan, A. Mohamed, and A. A. Hamid, "Pico hydropower (PHP) development in Malaysia: Potential, present status, barriers and future perspectives," *Renew. Sustain. Energy Rev.*, vol. 81, no. June, pp. 2796–2805, 2018, doi: 10.1016/j.rser.2017.06.084.
- [7] A. I. Siswantara, Warjito, Budiarmo, R. Harmadi, M. H. Gumelar, and D. Adanta, "Investigation of the  $\alpha$  angle's effect on the performance of an Archimedes turbine," *Energy Procedia*, vol. 156, pp. 458–462, 2019, doi: 10.1016/j.egypro.2018.11.084.
- [8] A. I. Siswantara, H. M. S. Gumelar, Budiarmo, R. Harmadi, Warjito, and D. Adanta, "Analysis of the Effects of Overflow Leakage Phenomenon on Archimedes Turbine Efficiency," *Proc. - 2018 4th Int. Conf. Sci. Technol. ICST 2018*, vol. 1, no. November, pp. 1–6, 2018, doi: 10.1109/ICSTC.2018.8528687.
- [9] G. Dellinger *et al.*, "Computational fluid dynamics modeling for the design of Archimedes Screw Generator," *Renew. Energy*, vol. 118, pp. 847–857, 2018, doi: 10.1016/j.renene.2017.10.093.
- [10] A. Židonis and G. A. Aggidis, "State of the art in numerical modelling of Pelton turbines," *Renew. Sustain. Energy Rev.*, vol. 45, pp. 135–144, 2015, doi: 10.1016/j.rser.2015.01.037.