

PAPER • OPEN ACCESS

## Numerical investigation on effect of blade shape for stream water wheel performance.

To cite this article: N F Yah *et al* 2018 *IOP Conf. Ser.: Mater. Sci. Eng.* **342** 012066

View the [article online](#) for updates and enhancements.

### Related content

- [New inverse method of centrifugal pump blade based on free form deformation](#)  
R H Zhang, M Guo, J H Yang *et al.*
- [Fabrication and high-speed characterization of SU-8 shrouded two-dimensional microimpellers](#)  
Atsushi Nakajima, Piljoong Kang, Nao Honda *et al.*
- [Study on the influence of back blade shape on the wear characteristics of centrifugal slurry pump](#)  
X Cai, S P Zhou and S Li

# Numerical investigation on effect of blade shape for stream water wheel performance.

N F Yah, A N Oumer, A A Aziz and I M Sahat

Faculty of Mechanical Engineering, Universiti Malaysia Pahang, 26600 Pekan Pahang.

E-mail: norfadilahyah@gmail.com

**Abstract.** Stream water wheels are one of the oldest and commonly used types of wheels for the production of energy. Moreover, they are economical, efficient and sustainable. However, few amounts of research works are available in the open literature. This paper aims to develop numerical model for investigation of the effect of blade shape on the performance of stream water wheel. The numerical model was simulated using Computational Fluid Dynamics (CFD) method and the developed model was validated by comparing the simulation results with experimental data obtained from literature. The performance of straight, curved type 1 and curved type 2 was observed and the power generated by each blade design was identified. The inlet velocity was set to 0.3 m/s static pressure outlet. The obtained results indicate that the highest power was generated by the Curved type 2 compared to straight blade and curved type 1. From the CFD result, Curved type 1 was able to generate 0.073 Watt while Curved type 2 generate 0.064 Watt. The result obtained were consistent with the experiment result hence can be used the numerical model as a guide to numerically predict the water wheel performance

## 1. Introduction

Renewable energy is the cleanest energy resources compared to fossil fuel and nuclear energy. Hydro power is one of the renewable energy resources which is expected to play a prominent role in power generation due the decreasing of global fossil fuels sources and supplying about 19% of electricity worldwide. Hydropower energy sources consists of waves, tides, ocean currents, natural flow of water in rivers, or marine thermal gradients [1]. Where it is not feasible to extend the main grid to Malaysia's remote rural communities, the decentralized generation of hydroelectricity can contribute significantly to improving the economic conditions of the rural populations of Malaysia. Hence, small scale hydro power plants may provide cost-effective energy alternatives to grid extension or isolated diesel mini-grids in rural areas of Malaysia. With hilly topography running almost the entire length and width of the country, and abundant number of streams flowing to foothills, Malaysia has a lot of small-hydro potential [2]. Basically, Small scale hydro power plant mainly equipped with reservoir or irrigation canal, governor, turbine and generator. Turbine is the heart of the system since it becomes the part which hydraulic energy will be converted to the mechanical energy.

Turbine broadly classified into positive displacement and dynamic categories. Basically, positive displacement turbines consist of small devices especially used for volume flow measurement, while dynamic turbine range from tiny to huge and used for flow measurement and power production. Dynamic turbines consist of impulse turbines and reaction turbines. Impulse turbines produced power

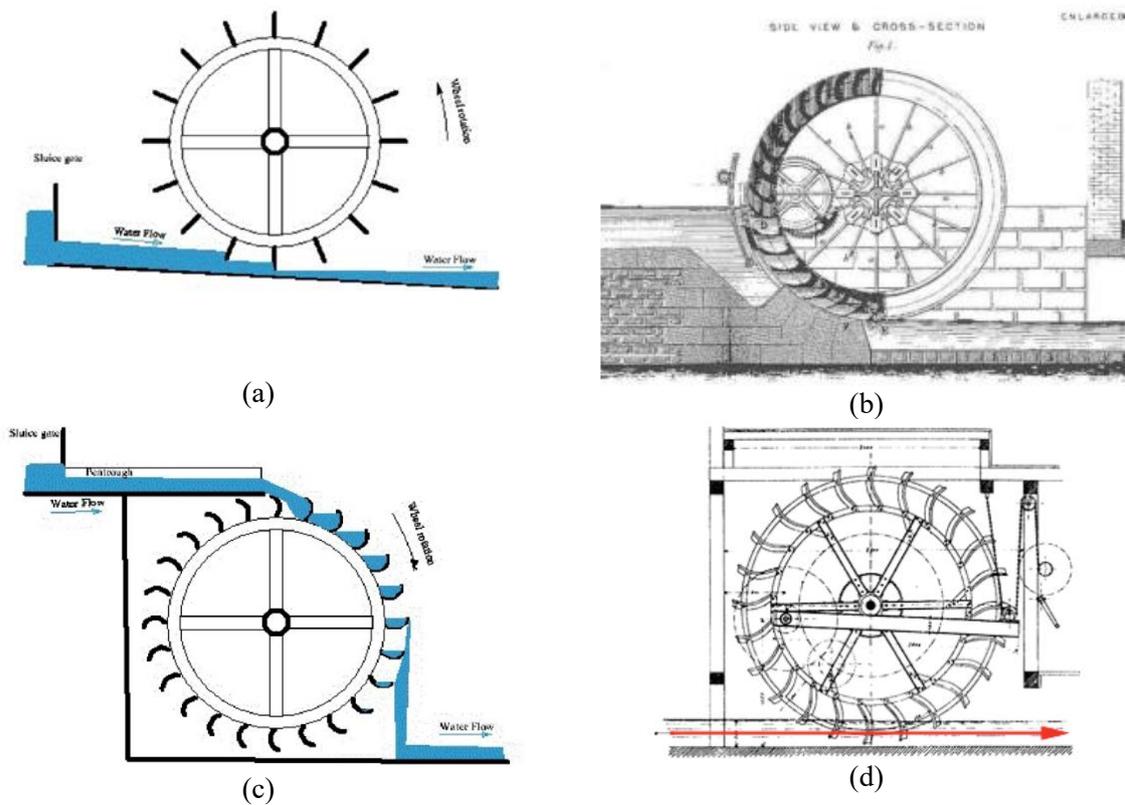


by the change in momentum due to the change of velocity direction of fluid flow. Impulsive force generated from the change of momentum will rotate the turbine runner. Therefore, the turbine are not necessarily to be immersed in the fluid since it is not driven by the pressure change in the fluid surrounding the runner. Reaction turbines are driven by the pressure change of fluid surrounding the runner that will cause it to rotate. Since the power generation depends on the pressure change of fluid, so the turbine need to be immersed in the fluid to make it functioning. Turbine selection process will consider the available head and flow on the site. For this study, the water wheel will be considered since the application is for the ultra-low head.

Water wheel are preferable energy converter for the small hydropower generation especially pico hydropower due to its ability to extract energy from low head water resources. The water wheel are believed to be originated in someplace at Mediterranean around 50 to 100 B.C [3]. It is classified as one of the oldest water turbine that has been used since antiquity until the introduction of high-pressure steam engines at the end of 18th century. Although this technology are invented since pre steam era, but the researcher and engineer nowadays are interested to study this hydraulic energy converter. This simple ancient machine that can be made of wood or steel with equally spaced blades or paddles around its circumference. It uses the falling or flowing water which flow and strikes surfaces of the blades. The impulsive force of water will rotates the wheel and generated torque will be transmitted to machinery via the shaft of the wheel [4]. Water wheels are widely used not only for mechanical power sources in flour and mineral mills, textile and tool making devices, but also used to generate electricity. Main advantages of water wheel compared to steam engines and turbines were their comparatively low cost and high efficiencies for a wide range of flow rates respectively [5].

There are four main type of water wheel which are undershot, breastshot, overshot and stream water wheel. They are characterized based on different level of water flow through the wheel and head required to operate. The first and believed as an oldest type is undershot water wheel. Some undershot wheel such as Zuppinger wheel employs only the potential energy of the water as the driving force. The idea of utilizing potential energy instead of kinetic energy of the low head water in rivers was noticed by a French engineer, Poncelet [5]. Undershot water wheel could operate in very small head differences of 0.5m to 2.5m and large flow volume ranging from 0.5 to 0.95 m<sup>3</sup>/s per m width. The water enters the wheel's blade below its axis. This water wheel utilized wherever a swiftly flowing river is available. In 19<sup>th</sup> century, the efficiency is around 25 % and was further developed until the efficiency can reached 70 % by Poncelet [1]. Figure 1 (a) below shows the undershot water wheel.

Breastshot water wheels were developed for head differences of 1.5 to 4m. Water enters the wheel at the level approximately to the level of wheel axis. The principal driving force of this water wheel is the weight of the water enclosed in the blade cell [2]. Figure 1 (b) below shows the breastshot water wheel. Overshot water wheel also known as the backshot water wheel was constructed to utilize head differences of 2.5m to 10m and flow rates of 0.1 to 0.2 m<sup>3</sup>/s per m width. As can be seen from Figure 1(c), water enters the wheel bucket from the top and the different weight of the water on other side causing the rotation of the vertically mounted wheel. Therefore, the buckets are designed in a way so that the water from above can enter each bucket at its natural angle of fall. The wheel design have utilized the force from falling water and weight of water collected in the bucket in order to rotate the wheel. But, the limitation is the head of water must be greater than the wheel diameter so that the water can be channelled from the top. This type of water wheel is fit for mountainous and hilly countries because its mechanical power is based on the wheel physical size and the available head [3]. Therefore, it requires the construction of dam, millpond and waterways. Stream wheels only utilizes kinetic energy of flowing water into mechanical energy. They are cost effective since only small construction works required to install the water wheel. Figure 1 (d) shows the stream water wheels. This paper aimed to study the effect of blade design and immersed depth ratio on the performance of the stream water wheel.

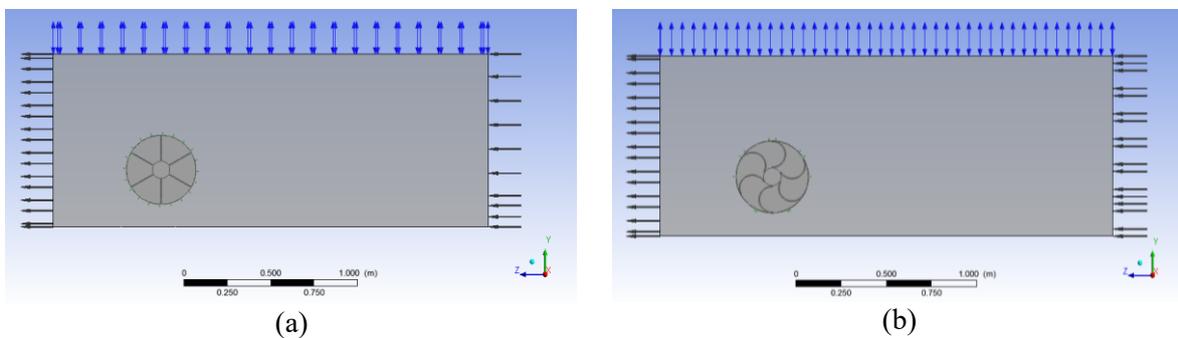


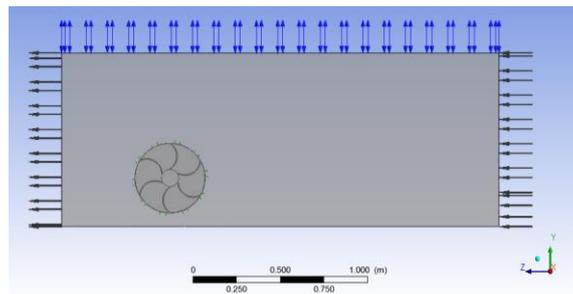
**Figure 1.** (a) Undershot water wheel (b) Breastshot water wheel (c) Overshot water wheel (d) Stream water wheel.

This paper aimed to study the effect of paddle number and immersed radius ratio on the stream wheel performance. The maximum powers of each paddle number and each immersed radius ratio were compared.

**2. Physical Domain**

The blade shape included straight, curved type 1 and curved type 2 with six number of blades. The schematic diagram of each blade design are presented in Figure 1 below.





(c)

**Figure 2.** Schematic diagram of the (a) Straight blade (b) Curved type 1 (c) Curved type 2.

The turbine is immersed in the flowing fluid (river) which is assumed to have constant flow velocity. The river flow field is considered to be incompressible, non-isothermal and behaves as Newtonian fluid (neglecting the effects of slurries inside the river). Therefore, the general forms of the continuity and momentum equations can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j) = 0 \quad (1)$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_i U_j) = - \frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + S_M \quad (2)$$

Where  $\rho$  is the density of water,  $U_i$  and  $U_j$  is the velocity vectors in direction of  $i$  and  $j$ ,  $p'$  is the modified pressure,  $\mu_{eff}$  is the effective viscosity accounting for turbulence. The term  $S_M$  represents the sum of body forces. The  $k$ - $\varepsilon$  model, like the zero equation model, is based on the eddy viscosity concept, so that:

$$\mu_{eff} = \mu + \mu_t \quad (3)$$

Where  $\mu_t$  is the turbulence viscosity. The  $k$ - $\varepsilon$  model assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (4)$$

Where  $\mu_t$  is a constant.

Torque and power is the essential parameter which need to be identified in order to evaluate the performance of the water wheel. From the experiment and numerical analysis, torque will be identified based on the different approach. During experiment, torque will be identified through the value of rotational speed,  $n$ . While, in numerical analysis the value of torque will be calculated by:

$$T = F_z \times r \quad (5)$$

$$P = T \times \omega \quad (6)$$

$$\omega = 2\pi n / 60 \quad (7)$$

Where  $T$  is the Torque in  $x$  axis direction of rotation in Nm,  $F_z$  is the force acted on the blade in  $z$  direction in N,  $r$  is the radius of the water wheel in m,  $P$  is the power generated by the water wheel in Watt and  $\omega$  is rotational speed in rev/s. The analysis of change in performance at different configuration

is performed by calculation the relative percentage in the interested variables. Percentage change of the variable on interest is calculated by Eq 8.

$$\text{Percentage change} = \frac{\text{Modified case} - \text{Reference case}}{\text{Reference case}} \times 100\% \quad (8)$$

In this study, a three dimensional with transient flow analysis was established by using Ansys CFX 15.0. In order to simulate the flow of the open channel flow, the uniform model of an Euler-Euler simulation was selected. In Ansys CFX, multiphase flow model can be applied with combination of Volume Fraction method. The basic equation used are based on mass, momentum and volume conservation. The standard k-epsilon model was adopted as the turbulent flow model, and the standard wall function was used to handle regions near wall surfaces.  $k$  is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity [10].

### 2.1 Geometry

The geometry of the water wheel was modelled in CAD software (Solidworks) and hence converted into readable file in Ansys such as Parasolid and STEP files. The dimension of the study from Anurat et al was used hence the water wheel is 0.2 m radius and each paddle is 0.1 m (W) x 0.15 m (H). The model was tested in 0.3 m (W) x 0.35 m (H) x 2.5 m (L) water channel with 0.3 m/s of water velocity [9].

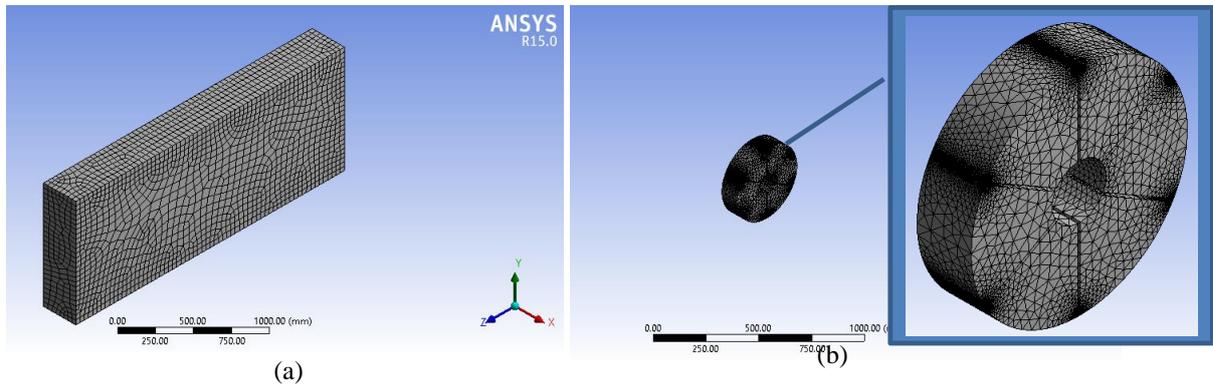
### 3. Boundary Conditions and Meshing

The mesh generation was done by using Ansys-ICEM CFD. This software is not only effectively able to mesh large and complex models but also can provide advanced size function of the grid. Basically, computational mesh are classified into structured mesh and unstructured mesh [11]. Structured meshes are not preferred for complex geometries but it is more efficient in terms of accuracy, CPU time and in most cases preferable. Unstructured mesh are preferable for complex geometries but the size difference between two adjacent elements can be large thus affect its numerical accuracy.

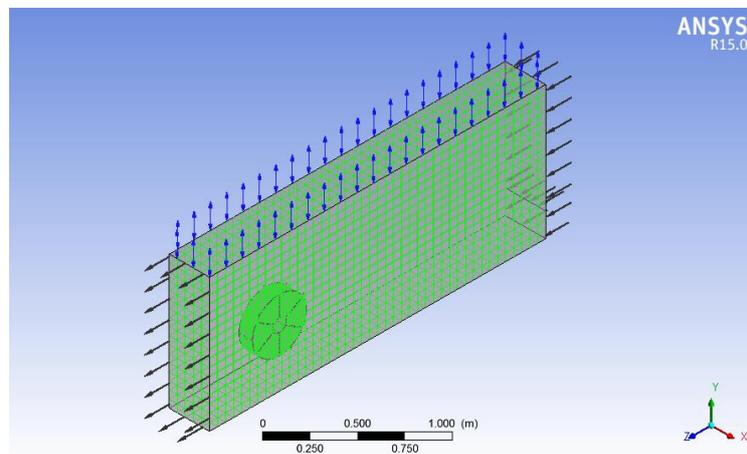
In this study, the simulation domain was divided into two parts; rotating and stationary parts. The stationary domain represents the domain for control surfaces of upstream and downstream region of the water channel. The rotating (turbine blade) domain meshed with advanced size function “proximity and curvature” and consist mainly of tetrahedral elements. The stationary domain which represent the upstream and downstream regions were meshed with advanced size function turned off and consist mostly hexahedral elements with some tetrahedral, prismatic and wedges element type. *Figure 3* represent the mesh for both stationary and rotating domain.

Analysis type was selected as transient analysis. There are two fluid domains were created, which is representing the rotating and stationary domains. The region of the turbine and water was separated into two different domain and mesh types to enable the rotating frame of reference used in the CFX [12][8]. The pressure around the turbine runner is the atmospheric pressure which is equal to 1 atm. The k-epsilon model was adopted since it has proven to be stable and numerically robust and has a well-established regime of predictive capability. This model are suitable for flows with boundary layer separation, flows with sudden changes in the mean strain rate, flows in rotating fluids, and flows over curved surfaces [13].

The inlet of water flow was set at the level depth of 0.18 m with the velocity of 0.3 m/s. In order to separate the water and air interface, volume fraction of the water was set to 1 and 0 for air. Water wheel was immersed in the water region with two different depth ratio which is 0.5. The complete setup was represented in the **Figure 4**.

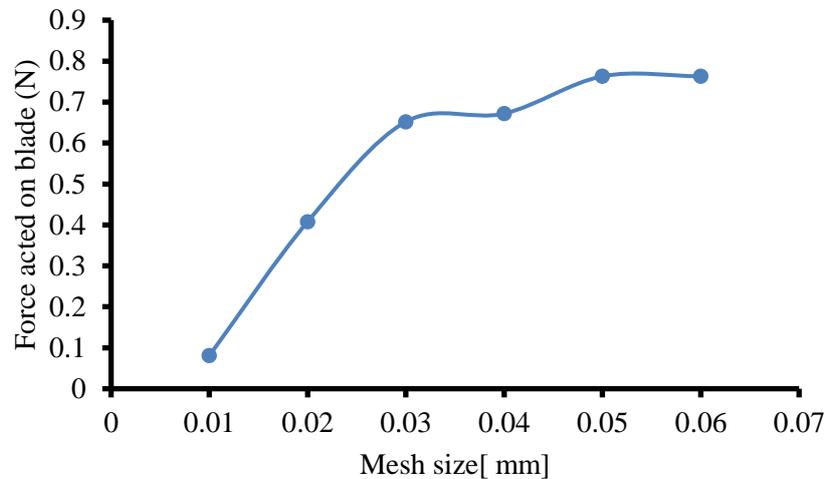


**Figure 3.** Meshing for (a) stationary and (b) rotating domain.



**Figure 4.** Complete setup for stationary and rotating domain.

Figure 5 below was plotted to represent the mesh independency test based on the mesh size of the both rotating and stationary domain. The force acted on the blade was numerically calculated based on the changes of the mesh sizes. It is observed that the mesh size at 0.05 and 0.06 have no changes on the force acted on the blade and the value of force is closer to experimental value. Therefore, the mesh size of 0.06 was predicted as a stable mesh size for the numerical study.



**Figure 5.** Mesh independency test for rotating and stationary domain.

The model was validated through comparison with the experimental result. Hence, the numerical study on the water wheel with different blade shape was conducted by applying the same numerical model. From the study, the torque and power generated from the flow was numerically calculated. Both of the model was tested with 0.5 immersed depth as this ratio was experimentally and numerically proven to be the optimum immersed depth ratio.



**Figure 6.** Configuration of different radius for curved type 1 and curved type 2.

#### 4. Result and Discussion

The result in this part consist of two part which is the validation of the numerical study by comparison between numerical and experiment result. Another part is the study of the blade shape by comparing the performance of the curved type blade with the straight blade.

##### 4.1. Model Validation

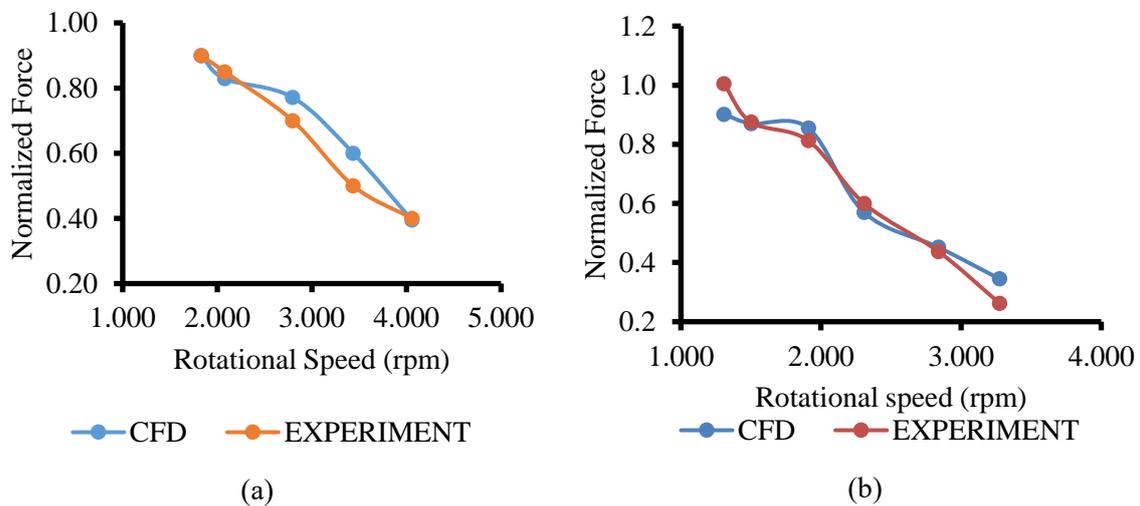
Validation process of the numerical model was conducted in order to validate the accuracy of the model used. Therefore, a numerical model was constructed based on the study proposed by the Nishi et al [14]. In this simulation, rotational speed was varied as experiment and force acted on the blade was numerically calculated. Immersed depth ratio of the water wheel was set to 0.5 and 0.75 and performance was observed [9]. The force acted in z-directions was plotted in the figures below. As can be seen in the Figure 7 (a), force generated on the blade surface at the immersed depth ratio of 0.5 was decreasing with the increasing of the rotational speed. While Figure 10 (b) plots the force generated on the blade for 0.75 immersed depth ratio. The force was responsible to generate the torque and power.

In experiment, the power generated from the water wheel at the flow velocity of 0.3 m/s was increased up to a maximum and then it decreases. The highest value of power generated at the 2.797

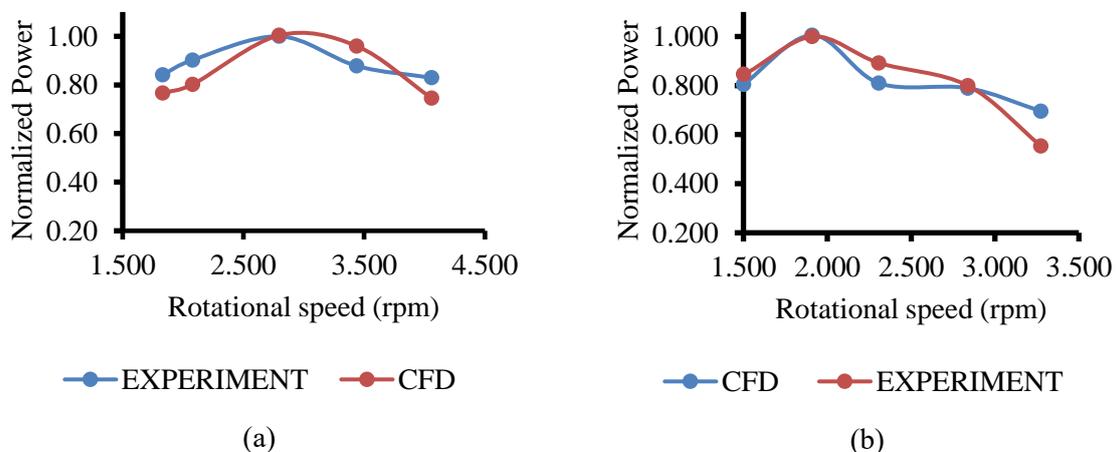
rpm and torque load of 0.14 Nm. After this value, which is equivalent to 0.041 Watt the power generated was decreased slightly although with increasing of rotational speed.

In CFD study, the inlet flow velocity was set to 0.3 m/s and the rotational speed of the water wheel was changed as same as in the rotational speed varied in the experiment. During 0.5 immersed depth ratio, power generated from the water wheel initially increase until a maximum value of 0.045 Watt at the 2.797 rpm and was slightly decreases although with the increasing of rotational speed. The result of generated power for both experiment and CFD was compared and presented in Figure 11. Therefore, the 0.5 immersed depth ratio performance predicted in the CFD study are likely same as the experimental results obtained in the literature and the highest power generated at the same rotational speed.

The normalized power generated from the numerical calculation and experiment for 0.75 immersed depth ratio was plotted in the Figure 8 (b). From the previous experiment result, the value of normalized power initially increase to maximum value and slightly decrease to minimum. Maximum power was generated at the 1.91 rpm and torque load of 0.16 Nm. Maximum power is 0.03 Watt and the generated power start to decrease after this maximum value. In CFD study, power generated also slightly increase to 0.034 Watt at 1.91 rpm and rapidly decrease to minimum value.



**Figure 7.** Generated force for (a) 0.5 immersed depth ratio (b) 0.75 immersed depth ratio.



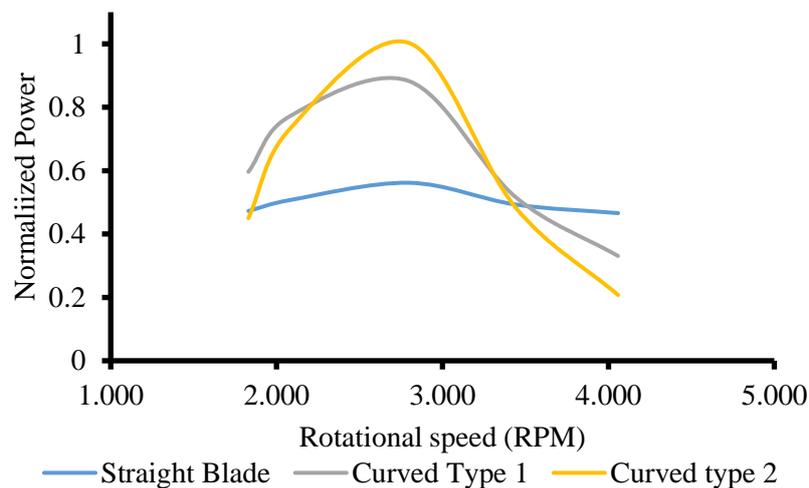
**Figure 8.** Comparison of experimental and numerical value of normalized power generated by straight blade water wheel with immersed depth ratio (a) 0.5 (b) 0.75.

Hence, the numerical model proposed in this study are acceptable since the performance of the water wheel in the experimental and numerical study are presenting the same trends. The average percentage of the error are around 9.3 %. Therefore, the model are validated since it able to predict the maximum power generated by the water wheel with acceptable percentage of error.

#### 4.2. Effect of Blade Shape

The performance of the water wheel with the straight and curved blade was studied by performing computational fluid dynamics with the same numerical model. The rotational speed was changed and the generated torque and power of the models was observed. Figure represents the changes of power generated by the water wheel based on the different rotational speed at the optimum immersed depth ratio of 0.5 [9]. It can be observed that the performance of each of the water wheel with curved blade type 1 and type 2 have increasing compared to the straight blade.

Previously, straight blade water wheel produces 0.041 Watt experimentally and 0.045 Watt numerically. From the study, it is observed that the curved type 2 has generate highest power output which is 0.073 Watt compared to 0.064 Watt generated by curved type 1.



**Figure 9.** The performance of straight, curved type 1 and curved type 2 based on the different rotational speed (rpm).

## 5. Conclusion

The performance of straight blade water wheel was numerically and experimentally investigated. From the simulation, result obtained was compared with the previously conducted experiment in order to validate the computational model used. The maximum power generated was 0.045 Watt which is small error of 9.3 % compared to the power generated during experiment. Therefore, the computational model used are validated to be appropriate prediction of the water wheel performance with different blade shape. Another two blade shape was numerically investigated which is curved blade with different curvature. From the simulation, curved blade type 2 has numerically generate highest power which is 0.073 Watt. Therefore, it can be concluded that curved blade are better than straight blade in terms of power generation.

This research examined the potential of utilizing very low head sites for small hydropower technology. Stream water wheel is a suitable water turbine for utilizing the water power from open channel flow. The outcome of this study contribute to the existing knowledge on very low head hydropower and the performance characteristics of hydraulic turbines with similar working principle.

### Acknowledgement

The authors would like to thank Universiti Malaysia Pahang for providing necessary assistance and Ministry of Higher Education for providing financial support under FRGS project number RDU140127.

### References

- [1] Güney MS and Kaygusuz K 2010 Hydrokinetic energy conversion systems: A technology status review *Renew. Sustain. Energy Rev.* **14** (9) pp. 2996–3004
- [2] Liu L, Masera H and Esser D 2013 World Small Hydropower Development Report
- [3] Reynolds TS 2002 *Stronger than a hundred men: a history of the vertical water wheel*
- [4] Farret FA 2006 *Integration of alternative sources of energy* John Wiley & Sons.
- [5] Müller G 1983 Water Wheels As a Power Source pp. 1–9
- [6] Turnock S, Muller G and Nicholls-Lee R 2007 Development of a floating tidal energy system suitable for use in shallow water
- [7] Cengel JM and Cimbala YA 2006 *Fluid Mechanics Fundamentals and Applications* The McGraw Hill Companies
- [8] Barstadm LF 2012 CFD Analysis of a Pelton Turbine
- [9] Tevata A and Inprasit C 2011 The Effect of Paddle Number and Immersed Radius Ratio on Water Wheel Performance *Energy Procedia* **9** pp 359–365
- [10] ANSYS CFX-Solver Theory Guide 2013 **15317** pp 724–746
- [11] Al Sam A 2010 Water wheel CFD simulations
- [12] Nishi Y, Inagaki T, Li T, Omiya R and Hatano K 2014 Research on the flow field of undershot cross-flow water turbines using experiments and numerical analysis *IOP Conf. Ser. Earth Environ. Sci.* **22** (6) p 062006
- [13] Canonsburg TD 2009 ANSYS CFX-Solver Modeling Guide **15317** pp 724–746
- [14] Nishi Y, Inagaki T, Li Y and Hatano K 2015 Study on an Undershot Cross-Flow Water Turbine with Straight Blades