The use of CFD as the design tool for designing a gravitational water vortex turbine

Nur Nazartul Ainna Nazarudin¹, Ernie Mat Tokit¹,², Mohd Afzanizam Mohd Rosli¹,², Fatimah Al Zahrah Mohd Sa‘at¹,²,³, Safarudin Gazali Herawan³,⁴, Said Ibnu Abi Syahputra⁴

¹Fakulti Kejuruteraan Mekanikal (FKM), Universiti Teknikal Malaysia Melaka (UTeM)
²Centre for Advanced Research on Energy (CARe), Fakulti Kejuruteraan Mekanikal, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia.
³Centre of Excellence Geopolymer and Green Technology (CeGeoGTech), Universiti Malaysia Perlis (UniMAP), 01000 Kangar, Perlis, Malaysia
⁴Industrial Engineering Department, Faculty of Engineering, Bina Nusantara University, Jakarta, Indonesia 11480

*Corresponding Email: safarudin.gazali@binus.edu

Abstract. Finite Element Analysis is commonly used for product designer to visualize the structural analysis of a product. Not much works done by these designers using Computational Fluid Dynamic, CFD due to the complexity of the application. In this study, the dynamic of the water flow within the turbine had been predicted as the preliminary design steps. Several significant parameters had been reviewed and tested using CFD tool to give better insight of the important components of the turbine on its performance, in terms of the velocity, the pressure and the vorticity of the water. Three different velocities were used which was 5 m/s, 9.9 m/s and 13 m/s. The maximum outlet velocity that can be produced at various inlet velocity were predicted to be 17.09 m/s, 33.79 m/s, and 44.37 m/s. The pressure gradients were 159 kPa, 618 kPa, and 107 kPa. Meanwhile, the vorticities were 48.28 m/s, 91.66 m/s, and 120.5 m/s. Furthermore, the vorticity distribution was observed through the simulated work.

1. Introduction
Finite Element Analysis is the common method used to analyze the structural analyses of a product design. The structural analysis is always being paid with high attention when designing a product to understand further the structural mechanics problem. A previous study had evaluated the stress concentration for a unique temporomandibular joint prosthesis [1]. It is rare that the flow behavior within the turbine is being analyzed by the product designers since it involves complex solutions. Therefore, the dynamic flow within the designed product unintentionally being ignored.

Through Computational Fluid Dynamics (CFD), the product designers can observe the changes of the dynamic flow within the turbine at divergent parameters around or within the turbine. By doing this, large parametric studies can be examined and the optimum dimension of the product design can be determined. Several sets of costly physical testing can also be prevented. Besides, CFD gives an insight of the visual flow inside the turbine to highlight problem areas that
may exist.

The CFD had been used in investigating the performance of the wind lens [2] at different angles. Another previous study used CFD to predict the airflow and temperature of rack cooling performance in the A2 Class Data Center. The predicted simulation result is then being used to propose the new rack layout recommendation for better thermal distribution [3]. Meanwhile, a wind turbine is designed to operate at a low Reynolds number with the help of CFD [4]. The same area of study about the wind turbine performance is also being numerically investigated in terms of the turbine power coefficient over the 2D hill [5]. Another utilization of CFD method is applied in the study of the novel wall-mounted cavity having a three-dimensional shape [6]. Apart from the CFD approach, a dynamic modeling had been used to simulate the transient performance of a heat pump system that is being utilized to develop the smart-controlled thermal energy system [7].

The gravitation water vortex power plant is a type of micro hydro vortex turbine system which is capable of converting energy in a moving fluid to rotational energy using a low hydraulic head of 0.7–3 meters. The technology is based on a round basin with a central drain. Above the drain the water forms a stable line vortex which drives a water turbine. In this study, a gravitational water vortex turbine (GWVT) had been designed to generate electrical power. The energy is generated using a gravitational water vortex which is formed when the water drains from the rooftop. The GWVT basically consist of a runner and a basin with a low head range of 0.7 m to 0.2 m. The normal top part of the turbine is free to the air [8] and this turbine is normally utilized in the river stream. However, this GWVT is designed with confined space which is not being investigated in any work before. It is important to know the effect of confined space on the formation of vorticity and the velocity distribution inside the turbine. Hence, preliminary design analysis of the gravitational vortex turbine is significant as a secondary tool for flow dynamics within the designed product, instead of structural analysis.

The GWVT itself can generate electricity without using any additional devices. However, the presence of the turbine helps a lot in improving the performance of the GWVT. The speed of the turbine mostly influences the circulation of the water vortex at first. Then, the strength of water vortex starts to weaken when an operating load is applied. At this moment, the water tends to flow in the axial direction. Thus, the aid from the turbine blade could extract the angular momentum better to produce a better efficiency [9].

To analyze further towards the design concept, several significant parameters had been investigated and had been addressed in some other works. These include the inlet geometry, basin configuration, runner to basin height ratio [10] and other parameters as explained later.

The investigation of the inlet geometry revealed that the triangular inlet gave the best quality of the water vortex as it resulted in a very less force along the radial direction. The radial direction could reduce the efficiency and the durability of the turbine [11]. With the help of a guided plate, the flow at the basin center created a water vortex [12]. Alternatively, the basin configuration also helped in the formation of a water vortex turbine. The comparison made between cylindrical and conical shape basin showed that the vortex maximum vorticity strength occurred at 50 mm radius of the conical basin [13]. Other numerical result showed that the concave design gave out 1.81 m/s of outlet speed while 1.12 m/s for convex design [14]. Thus, these findings showed that the shape of the basin was important to gain a maximum velocity for GWVT. Another numerical result had shown the effect of the inclination angle of rectangular passage on the velocity vector and the flow field. The basin had a 1 m upper diameter, 0.4 m outlet diameter with the different angles of rectangular passage attached as the inlet for the basin. The inlet velocity was 3 m/s for three various angles; 30°, 45°, and 60°. The maximum velocity achieved was 10.3 m/s, 10.88 m/s and 10.96 m/s respectively. Therefore, the rectangular passage with 60° inclination angle produced better result in terms of maximum produced velocity [15].

Apart from that, the best position for a runner was about 60% to 70% of height from the top of the basin. For the conical basin, the maximum velocity gained was 0.6 m/s while for the cylindrical basin, the maximum velocity was 0.525 m/s [16]. Another experiment also had been conducted to ensure the position of a runner in a conical basin. The total height of the basin used was 1300 mm. The runner had been tested at three different heights, 440 mm, 740 mm and 840 mm height from the top and the velocity obtained was 0.51 m/s, 0.61 m/s and 0.64 m/s respectively [17].

Other factor that is needed to be considered is the number and the design of the blade which also affect the performance of the vortex turbine. In Thailand, tests with different percentage of
baffled plate were conducted. The tests conducted with the observation based on the torque produced and its efficiency at 0.04 m$^3$/s, 0.05 m$^3$/s and 0.06 m$^3$/s mass flow rate. Throughout the experiment, the turbine with 5 blades and 50% baffled plate gave out the highest torque with average value of 37.41 Nm and efficiency at three different mass flow rates with average percentage 32.79% as compared to the normal turbine which had 33.93 Nm with 31.49% average percentage of efficiency [18].

In another experimental work, it was found that the blade length affected the vortex turbine hydraulic efficiency. The type of blade used for this experiment was cross flow blade. The maximum hydraulic efficiency, 42.1%, was found when the turbine outer diameter was 0.035 m, 0.027 m blade length, 0.0075 m inner diameter, 0.07 m blade height, 0.0025 m blade thickness with three number of blades [19].

To improve further the turbine’s performance, a booster runner is added. It is positioned lower than the main runner and coupled in the same shaft, closed to the outlet. Based on the numerical analysis, the maximum efficiency increased from 76.03% to 78.65% when a booster runner was used [20]. Another numerical analysis had been done on two-stage turbine using Savonius blade and three-stage turbine using cross blade and twisted blade. This study found that the power generation at the bottom stage near the outlet was strongly supported by hydraulic drop because of the vortex generation below it that caused from the lower stage [21].

The flow analysis within the turbine can also be discussed in terms of the vortex. The vortex distortion of the lower runner was adversely affected due to the operational of the top stage. Besides, the top runner blade profile must be designed for minimum vortex distortion. This was done by using the cross flow blade for the top, and the tilted blades on the vertical plane for the bottom part. However, the water load above it has aided the lower runner to boost the torque and power to improve the performance of the lower runner [22]. Furthermore, it is suggested to vary the rotor diameter to basin diameter. Experimental result showed that the rotor to the basin diameter ratio is 0.6, which can produce an average value of 3.85 W output power [23]. Moreover, the runner profile also affected the turbine performance. A work done to investigate the effect of the runner profile showed that the modified runner (3D type) worked better as compared to flat radial, paddle, centrifugal and modified form. The maximum efficiency was 24% which run at optimum rotational speed of 90 to 120 rpm [24]. As a result, the maximum power yielded in gravitational water vortex turbine was 14.5 W with 35.92% of efficiency while the small under shot water wheel had maximum power of 7.5 W with the efficiency of 13.96% [25].

2. Methodology
This section explain the detail methods used to carry out the work flow throughout this study.

2.1. Flowchart of the work
The GWVT is one of the significant components used to harvest the rooftop rainwater for electrical generation. The whole view of this system is shown in Figure 1 and this study focuses on the design of the turbine.

![Figure 1. Schematic view of the roof top rainwater harvesting system](image-url)
The flowchart of the study is shown in Figure 2. Figure 2 shows the diagrammatic representation of the whole work process before the GWVT being fabricated. The study scope is being highlighted through the enclosed area of the dash-circle.

2.2. Model of the domain
In this work, a flow simulation had been used as a preliminary design tool for a water gravitational turbine. The turbine had been modelled and scaled down based on previous work [17] to suit with the roof top water harvesting system. The top part is not free to the air surface but closed as confined area. Figure 3 shows the dimension of the turbine used in this study.

The fluid dynamics of flow for the designated gravitational turbine was solved using ANSYS Fluent. A steady model was used with water treated as incompressible fluid. The numerical calculations were all set with second order accuracy. The standard Navier Stokes Equations of flow were used. Convergence for all the governing equations was set at 1×10⁻⁶.

2.3. Boundary condition
The boundary condition sets the initial value at the basin inlet and outlet, and also the condition at the upper basin and conical wall. The boundary condition of the turbine model is depicted in Table 1 and the set-up water properties had been presented as in Table 2.

| Boundary Condition          | Value or surface |
|-----------------------------|------------------|
| Inlet velocity (Vin)        | 0.18 m/s         |
| Outlet pressure (Pout)      | 0 Pa             |
| Wall                        | Upper surface    |

| Properties                          | Value                      |
|-------------------------------------|----------------------------|
| Density                             | 1000 kg/m³                 |
| Dynamic viscosity at 25°C           | 8.9 x 10⁻⁴ Ns/m²           |
| Kinematic viscosity at 25°C         | 0.8927 x 10⁻⁶ m²/s         |

3. Results and Discussion
This section describes the distribution of velocity, pressure, and vorticity within the turbine basin.

3.1. Velocity distribution
Figure 4 shows the velocity distribution near the blade region displayed from the top view for water inlet velocity of 5 m/s, 9.9 m/s and 13 m/s. The increment of the water inlet velocity leads to higher velocity distribution near the runner. The tangential velocity near the basin wall is higher and becomes slower towards the basin center. As the water inlet velocity increases, the water velocity around the blade increases too. The increment of the maximum water velocity near the blade seemed to increase linearly as the water inlet velocity increases. The contour also shows that the tangential velocity near the basin wall is higher as compared to the radial velocity the basin center.
Figure 2. The flowchart of the work process

Figure 3. a) Front view, b) side view and c) top view of the model
Figure 4. Velocity contour near the blade for water velocity inlet of a) 5 m/s, b) 9.9 m/s and c) 13 m/s

Meanwhile, Figure 5 and Figure 6 show the velocity contour on the vertical plane and the top plane along the basin. The shape of the basin at certain inclination angle increases the tangential velocity near the basin wall and produces higher velocity towards the outlet of the basin. Maximum velocity increases at the outlet as the inlet water velocity increases too. This can be viewed as the outlet velocity reaches up to 17.09 m/s, 33.79 m/s, and 44.37 m/s respectively. Through the conservation of momentum, the direction of tangential velocity is departed as the water flows out to the basin outlet.

Figure 5. The velocity distribution along the vertical plane within the conical basin at a) 17.09 m/s, b) 33.79 m/s, and c) 44.37 m/s

Figure 7 shows the plotted water velocity along the crossed points as shown in Fig. 3. The velocity is nearly reaches 0 m/s near the basin axis. The velocity is not distributed evenly since the inlet geometry does not lies symmetrically. This causes the mass of water to become dense on certain part of the basin wall, and yet leads to higher velocity in that particular region. Hence, the uneven velocity distribution is predicted along the points.
Figure 6. The velocity contours along the plane for water inlet velocity of a) 5 m/s, b) 9.9 m/s, and c) 13 m/s

Figure 7. Velocity contour at different planes for water velocity inlet of a) 5 m/s, b) 9.9 m/s, and c) 13 m/s

3.2. Pressure distribution
Figure 8 and Figure 9 show the pressure distribution on the vertical plane along the basin. The movement of the fluid creates dynamic pressure. In accordance to Bernoulli principle, the increment and the decrement of the velocity and the pressure are interrelated. The fluid exerts the highest pressure near the wall and decreases towards the outlet of the basin. The exist of pressure gradient forces the fluid to accelerate and follows the curved path of the basin wall. The pressure drop is predicted at 159 kPa, 618 kPa, and 107 kPa as the inlet water increases at 5 m/s, 9.9 m/s, and 13 m/s.
Figure 8. Pressure distribution contours on the vertical plane at various water inlet velocity of a) 5 m/s, b) 9.9 m/s, and c) 13 m/s

Figure 9. Pressure distribution contours along the horizontal plane for water velocity inlet of a) 5 m/s, b) 9.9 m/s, and c) 13 m/s

3.3. Vorticity

There are several factors that had been identified to affect the vorticity of the water flow. One of it is the ratio of the outlet diameter to the diameter of basin, d/D which is in the range of 0.14 to 0.18. However, other previous work concluded that the ratio could be in the range of 0.2 to 0.35 to gain the best efficiency. For this model, the ratio of d/D is 0.25. Fig. 10 shows the velocity vector that is formed from the top view of the basin. This shows that there is a vortex formation inside the GWVT. The strongest vortex occurs at the axis of the basin and decreases as the fluid moves away from the center of the axis. This shows the linear relation of the vorticity with the radius of the basin. Contradict to the velocity contour, the vorticity shows a decrement pattern as the water flows to the basin outlet. The vorticity near the runner is weak as the water flow is obstructed by the blades. Through this analysis, it shows that the axis of the runner should be the same as the axis of the basin to ensure the blade runs at the highest rotation. The predicted vorticity are 48.28 m/s, 91.66 m/s, and 120.5 m/s.
4. Conclusions

In this study, the CFD method is used as a design tool for the gravitational water vortex turbine. The analysis of the velocity, pressure and the vorticity can be predicted and can be utilized as the preliminary prediction on the effect of the basin configuration. The maximum outlet velocity that can be produced at various inlet velocity are predicted to be 17.09 m/s, 33.79 m/s, and 44.37 m/s. The pressure gradients are 159 kPa, 618 kPa, and 107 kPa. Meanwhile, the vorticities are 48.28 m/s, 91.66 m/s, and 120.5 m/s. All the parameters are significant to determine the highest power that can be generated by the turbine.

Acknowledgements
We acknowledge Universiti Teknikal Malaysia Melaka (UTeM), Malaysia and our collaborator, Bina Nusantara University as a part of Bina Nusantara University’s International Research Grant entitled Development of Siphonic-Roof-Top Water Turbine for Green Technology with contract number: No.026/VR.RTT/IV/2020 and contract date: 6 April 2020 for encouraging and supporting this research work.

References

[1] Sharma M and Soni M 2020 A musculoskeletal Finite Element Study of a Unique and Customised Jaw Joint Prosthesis for the Asian Populace EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy 7(03) 351-358
[2] Takeyeldein M M, Lazim T M, Ishak I S, Nik Mohd N A R and Ali E A 2020 Wind Lens Performance Investigation at Low Wind Speed EVERGREEN 7(4) 481-8
[3] Wulandari D A, Akmal M and Gunawan Y 2020 Cooling Improvement of the IT Rack by Layout Rearrangement of the A2 Class Data Center Room: A Simulation Study EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy 7(4) 489-99
[4] Takeyeldein MM 2019 Wind turbine design using thin airfoil sd2030 EVERGREEN Joint Journal of Novel Carbon Resource Sciences & Green Asia Strategy 6(2) 114-23
[5] Ibrahim O M A M and Yoshida S 2018 Experimental and numerical studies of a horizontal axis wind turbine performance over a steep 2d hill EVERGREEN 5(3) 12-21
[6] Oka T, Handa T, Akagi F, Yamaguchi S, Aoki T, Yamabe K and Kihara Y 2017 Steady-state analysis of supersonic mixing enhanced by a three-dimensional cavity flow EVERGREEN 4(1) 44-51
[7] Ko J, Takata N, Thu K and Miyazaki T 2020 Dynamic modeling and validation of a carbon dioxide heat pump system EVERGREEN 7(2) 172-94
[8] Nishi Y and Inagaki T 2017 Performance and flow field of a gravitation vortex type water turbine International Journal of Rotating Machinery 2017
[9] Kueh TC, Beh S L, Ooi Y S and Rilling D G 2017 March. Experimental study to the influences of rotational speed and blade shape on water vortex turbine performance In Journal of Physics: Conference Series Vol. 822, No. 1, p. 012066 IOP Publishing

[10] Bajracharya T R, Shakya S R, Timilsina A B, Dhakal J, Neupane S, Gautam A and Sapkota A 2020 Effects of Geometrical Parameters in Gravitational Water Vortex Turbines with Conical Basin Journal of Renewable Energy 2020

[11] Thapa D, Mishra A and Sarath K S 2017 Effect of inlet geometry in the quality of vortex formed using vortex flow channel International Journal of Mechanical Engineering and Technology 8(5) 515-24

[12] Wanchat S, Suntivarakorn R, Wanchat S, Tonmit K and Kayanyiem P 2013 A parametric study of a gravitation vortex power plant In Advanced Materials Research Vol. 822, No. 1, p. 012066 IOP Publishing

[13] Dhakal S, Nakarmi S, Pun P, Thapa A B and Bajracharya T R 2014 Development and testing of runner and conical basin for gravitational water vortex power plant Journal of the Institute of Engineering 10(1) 140-8

[14] Sánchez A R, Del Río J A S, Muñoz A J G and Montoya J A P 2019 Numerical and experimental evaluation of concave and convex designs for gravitational water vortex turbine Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 64(1) 160-72

[15] Rehman W, Ijaz M and Munir A 2017 Designing of micro gravitational vortex turbine’s vortex pool In ASME Power Conference Vol. 57618, p. V002T12A002 American Society of Mechanical Engineers

[16] Dhakal S, Timilsina A B, Dhakal R, Fuyal D, Bajracharya T R, Pandit H P, Amatya N and Nakarmi A M 2015 Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant Renewable and Sustainable Energy Reviews 48 662-9

[17] Kayastha M, Raut P, Subedi N K, Ghising S T and Dhakal R 2019 CFD evaluation of performance of Gravitational Water Vortex Turbine at different runner positions In KEC Conference 2019

[18] Wichian P and Suntivarakorn R 2016 The effects of turbine baffle plates on the efficiency of water free vortex turbines Energy Procedia 100 198-202

[19] Rahman M, Hong T J, Tang R, Sung L L and Tamiri F B M 2016 Experimental study the effects of water pressure and turbine blade lengths & numbers on the model free vortex power generation system International Journal of Current Trends in Engineering & Research (IJCTER) 2(9) 13-7

[20] Gautam A, Sapkota A, Neupane S, Dhakal J, Timilsina A B and Shakya S 2016 Study on effect of adding booster runner in conical basin: gravitational water vortex power plant: a numerical and experimental approach In Proceedings of IOE Graduate Conference 107-13

[21] Cheema T A, Ullah R and Saleem A S 2019 Performance analysis of a two-stage gravitational water vortex turbine In IOP Conference Series: Earth and Environmental Science Vol. 291, No. 1, p. 012039 IOP Publishing

[22] Ullah R, Cheema T A, Saleem A S, Ahmad S M, Chattha J A and Park C W 2019 Performance analysis of multi-stage gravitational water vortex turbine Energy Conversion and Management 198 pp.11788

[23] Ullah R, Cheema T A, Saleem A S, Ahmad S M, Chattha J A and Park C W 2020 Preliminary experimental study on multi-stage gravitational water vortex turbine in a conical basin Renewable Energy 145 2516-29

[24] Haryadi, Subarjah A M and Sugianto 2020 November. Experimental study on 3D vortex gravitational turbine runner In AIP Conference Proceedings Vol. 2296, No. 1, p. 020025 AIP Publishing LLC

[25] Sritram P and Suntivarakorn R 2017 Comparative study of small hydropower turbine efficiency at low head water Energy Procedia 138 646-50