Flow Analysis of Turgo Impulse Turbine for Low Head Power Plant

Hnin Hnin Ei[*], **Myat Myat Soe**[**]

[*] *Department of Mechanical Engineering, Mandalay Technological University*
Mandalay, Myanmar
[**] *Department of Mechanical Engineering, Mandalay Technological University*
Mandalay, Myanmar

DOI: 10.29322/IJSRP.16.7.2018.p8037
http://dx.doi.org/10.29322/IJSRP.8.7.2018.p8037

Abstract- Hydro energy is widely used in the small and remote areas that require only a small amount of electricity. The Turgo turbine is used for pico hydro power generation. The initial cost of the turbine is low and is similar to Pelton wheel. The Turgo impulse turbine is known for its reliability and strength, and it can perform efficiently with the range of flows. The research was carried out in order to perform CFD analysis of Turgo turbine. The whole simulation was performed in ANSYX-CFX. The Computational Fluid Dynamics (CFD) method was used to determine the stream flow through the blade. The Turgo Turbine consists of the runner with 261.50 mm diameter and the diameter of the nozzle is 26.15 mm which inclination angle is 20 degree. The flow rate of this turbine is 436.9 liter /min, the head is 10 m and the total number of buckets are 20 buckets. The power and torque generated by the device is analyzed and compared with CFD simulation under the certain boundary conditions. The simulation uses the k-ε turbulent flow model. The buckets are designed according to hydrodynamic theory. In this research the power and torque are compared between theoretical and numerical at various jet speeds for Turgo impulse turbine.

Index Terms- Flow, Jet velocity, Power, Torque, Turgo turbine

• Introduction

The Turgo turbine is the impulse turbine. The Turgo turbine is similar to the Pelton but the jet strike the plane of the runner at an angle of typically from 10° to 30° so that the water enters the runner on one side and exits on the other. Therefore the flow rate is not limited by the discharged fluid interfering with the incoming jet (as is the case with Pelton turbines). As a consequence, a Turgo turbine can have a smaller diameter runner and it can rotate faster than a Pelton for an equivalent flow rate.

The Turgo turbines are used in medium head and high head of water between the head of 15 m to 300 m. Turgo turbine transforms kinetic energy of water jet to rotational energy with the help of a nozzle. The high speed of water jet directed on to the turbine blades and the turbine is rotated at high speed after striking the water to the turbine blades. Then the shaft is rotated and the electricity is generated in generator. Turgo turbine runner is same as Pelton runner but it is split in half. For the same power output the Turgo turbine runner is one half the diameters of the Pelton runner, and the specific speed is twice the Pelton runner.

The Turgo turbine can handle a greater water flow than the Pelton turbine because the exiting water doesn't interfere with buckets. If the number of jets are increased the specific speed of the turbine is also increased. This is widely used in pico hydro power plant because the cost of the turbine is low and it can easily manufacture in minimum cost. Turgo turbines are mostly used in rural areas electrification in pico hydro power plants.

The simulation is important before the actual operation of turbine as it helps to optimize the design according to the obtained values of flow velocities, pressure distribution and efficiency. The results obtained from CFD are of great interest since these findings can be used to minimize testing time and cost as well as to analyze failure conditions. The research was performed to calculate the power and torque by the computational method.

METHODOLOGY

The power supplied by a water jet depends upon the head and the rate of water flow. Therefore the available hydraulic power can be written in terms of head and flow.

$$\text{power supplied by the jet, } P = \rho g Q H \tag{1}$$

Where ρ is the density of the fluid, H is hydraulic head and Q is the flow rate. The jet diameter d_j can be calculated using

$$d_j = 0.545\sqrt{\frac{Q}{z_o\sqrt{H}}} \tag{2}$$

Therefore, the diameter of runner is

$$D = m\, d_j \tag{3}$$

The jet diameter plays an important role in the designing of the buckets of the Turgo turbine. The width of the bucket B is (1.68-2.34) d_j [9] , the length of the bucket L is (2.4-3.4) d_j [9]and the depth of the bucket H is (0.3-0.585) d_j [9] respectively.

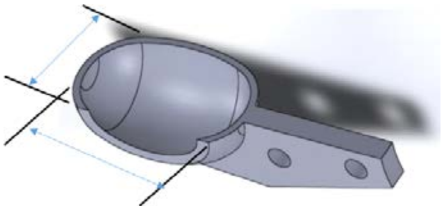


Figure 1. Bucket of Turgo turbine

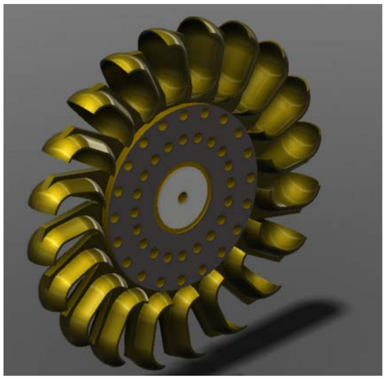


Figure 2. Buckets and runner of Turgo turbine

The jet velocity is determined from the net head as:

$$v_j = c_v\sqrt{2gH} \approx 0.98\sqrt{2gH} \tag{4}$$

Where c_v is the efficiency of the nozzle and generally 0.97 ~ 0.98 [12]. At optimum efficiency, the circumference or tangential velocity of the runner is connected with the jet velocity as:

$$U \approx (0.46 - 0.47)v_j \tag{5}$$

The power is the product of the angular speed at maximum power and the torque.

$$P = T\omega = \eta\rho gQH \tag{6}$$

NUMERICAL ANALYSIS

The CFD simulation involves numerical modelling and analysis of the flow in the Turgo runner, along with a preliminary design. To achieve this, the design of buckets and runner were initially designated according to the hydrodynamic theory. The CFD simulation starts with the creation of 3D study model. This study consists of the buckets, nozzle, casing, fluid part and rotating bodies. The materials considered in the CFD process are the fluid, solid, and rotating regions.

A. Geometric Modeling and Meshing

The flow domain is imported to Geometry sub-program by using the ANSYS Design Modeler. The flow domain consists of the fluid, the nozzle, the bucket, the casing and rotating region. The geometry of Turgo turbine is designed with the aid of the SolidWorks. The domains are separated into two parts such as the stationary components and rotating component respectively. 20 buckets were considered in this simulation and these buckets are taken as the rotating part. The nozzle and the casing are considered as the stationary parts.

The meshing was performed for both domains. The flow domain is meshed using the ANSYS Mesh sub-program ANSYS CFX Project. In the meshing, the size function is taken as curvature and the relevance center is considered as the medium. Figure 5 shows the meshed on buckets and nozzle and the numbers of nodes are 1045788 and the numbers of elements are 5393410. Figure 6 is the meshed detail on buckets. Figure 3 shows the geometry for rotating part and figure 4 describes the geometry for rotating and stationary part of the Turgo turbine.

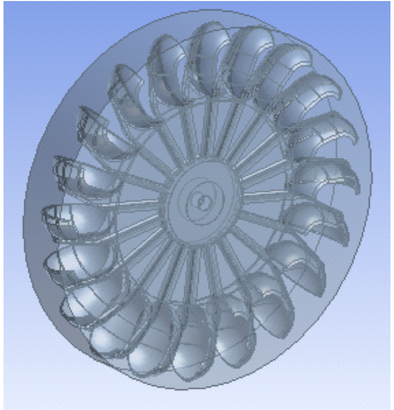


Figure 3. Geometry for rotating part

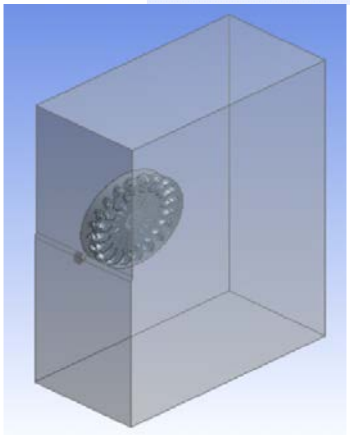
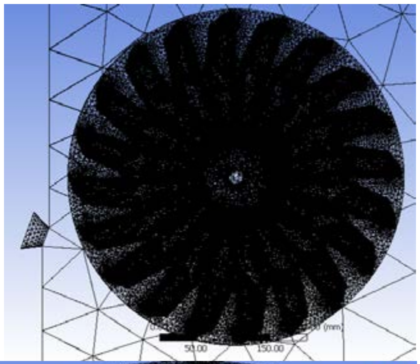


Figure 4. Geometry for rotating and stationary part



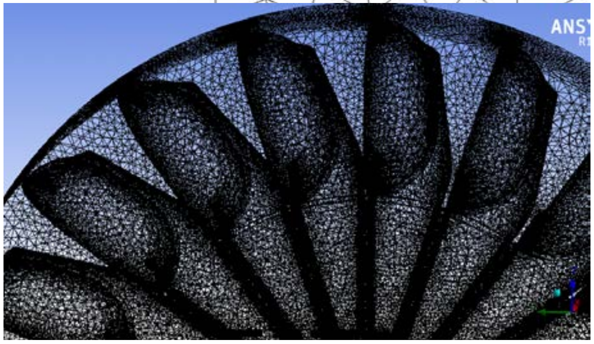


Figure 5. Meshed on buckets and nozzle

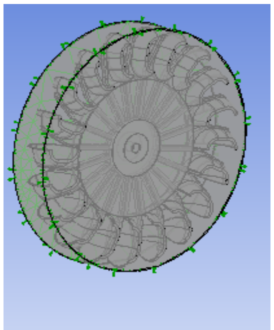
Figure 6. Meshed on buckets

Meshing on buckets and nozzle can be seen from figure 5 and figure 6 shows the detail mesh on buckets.

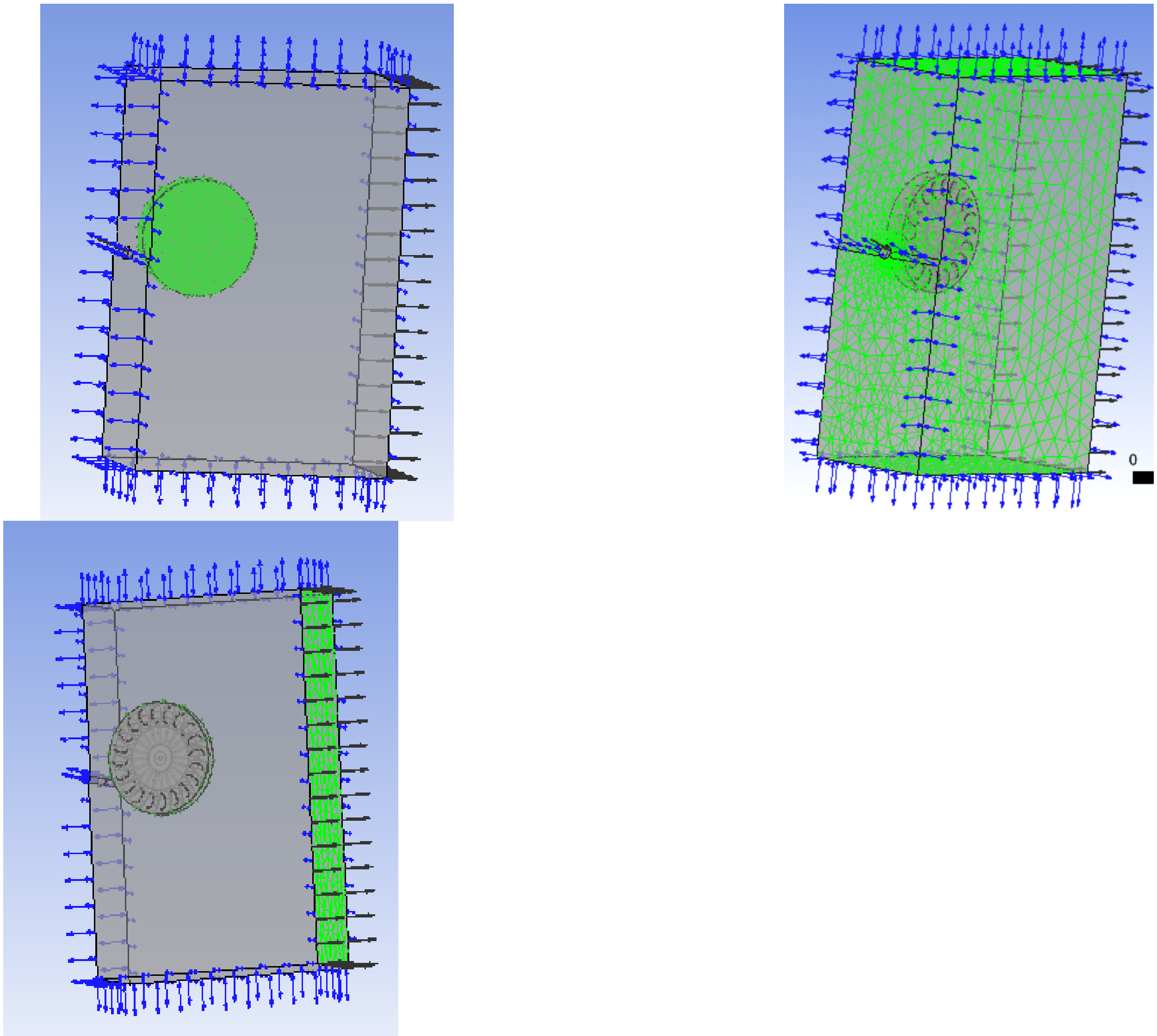
B. Boundary Conditions

The analysis was carried out at steady state. The reference pressure was kept to 1 atm for both domains and the fluid temperature is 25°C. The turbulence option is taken as k-ε and domain type is fluid domain. The angular velocity for rotating part is 48.29 rad/s and the mass flow rate is 7.28 kg/s. Domain motion was set to stationary for the nozzle and casing while rotating for runner and bucket with appropriate speed. The motion of the rotating domain was set in clockwise direction along positive x-axis direction.

In the input boundary parts, the water stream from the nozzle is injected into the rotating region. The inlet boundary condition for nozzle was 13.73 m/s. All buckets were set to wall with no slip condition. The faces at the stationary domain except nozzle are taken as the opening with relative pressure 1atm. But the back surface set as the outlet whereas the pressure outlet show 1 atm respectively. Figure 7 (a) shows the domain interface of rotating turbine.



(a) Domain interface



(b) Rotating (c) Opening (d) Outlet

Figure 7. Boundary Conditions

Figure 7 (b) illustrates the rotating part of Turgo turbine. The places around the stationary part such as front surface, top and bottom surface and the sides surface are taken as opening. But the back surface is taken as the outlet. Figure 7 (c) shows the opening part and (d) shows the outlet of the turbine

C. Simulation Domain

The pressure distribution, the velocity distribution and torque variation in the turbine bucket were analyzed after the simulation. The liquid flow in the rotating runner of an impulse turbine is complex and unsteady. The water emerges from the inlet nozzle and strikes the buckets. The inclination of jet angle is 20 degree. Figure 8 shows the streamlines of water jets discharged from the nozzles.

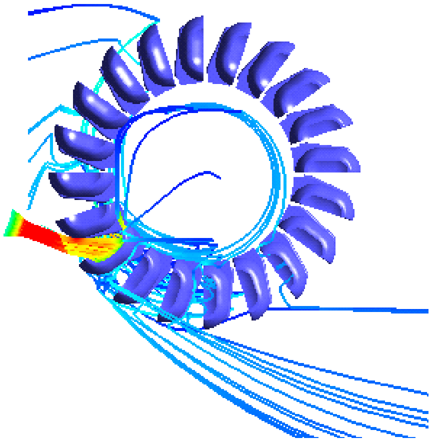
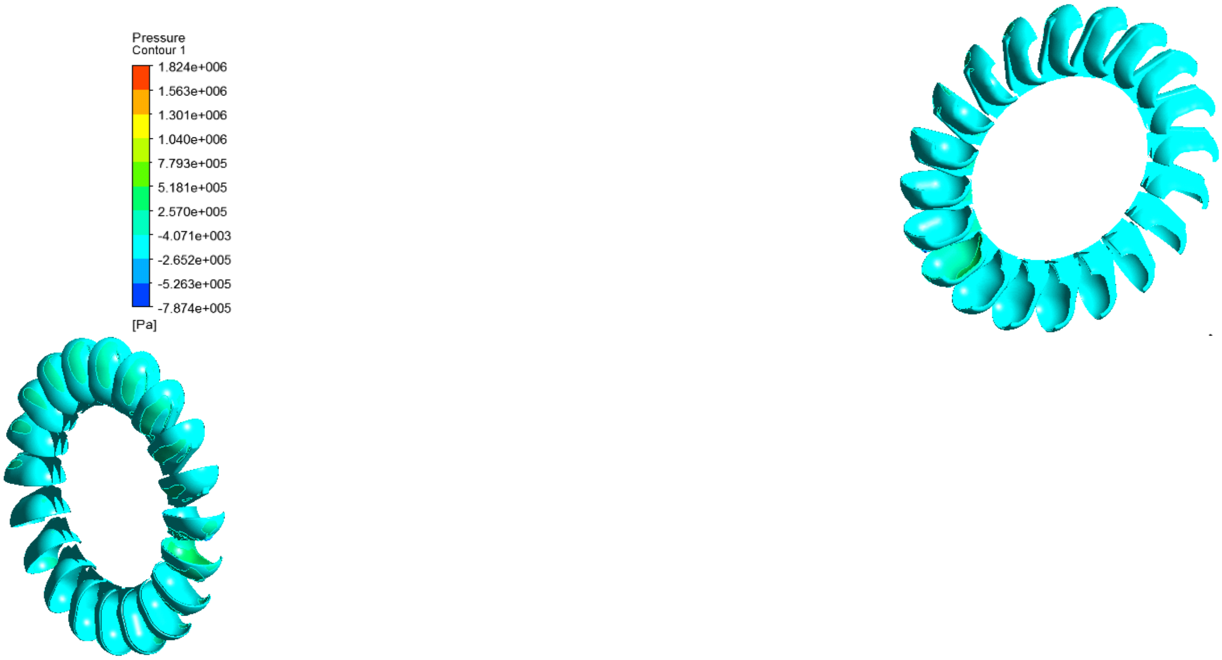


Figure 8. Streamline of water flow on the bucket

(i) Pressure distribution

The pressure distribution in the bucket was due to impact of high jet. The pressure is maximum at the place where the jet strike on the bucket with the value of 0.72 MPa.



(a) (b)

Figure 9. Pressure distribution on the buckets (a) front side (b) back side

Figure 9 (a) illustrates the pressure distribution on the buckets from the front side and figure 9 (b) also shows the pressure distribution on the back side of the buckets.

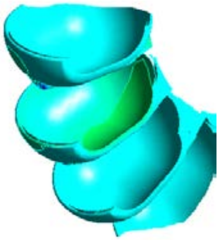


Figure 10. Pressure distribution on one bucket length

Figure 10 shows the pressure distribution along the bucket length where the jet strike. Since the inclination of the jet angle is 20, the pressure decrease at the tip of the bucket length and then it increase gradually. The maximum pressure occurs at the root of the bucket. From the simulation results, the pressure is maximum at the spot where the jet strikes the bucket, and then decreases towards the edges of the bucket. If the pressure increases, the force will increase proportionally and the bucket can be more failure.

(ii) Velocity distribution

Figure 11 shows the velocity distribution through the buckets. The velocity is maximum when the water emerge from the nozzle and then it will decrease when it strike the bucket.

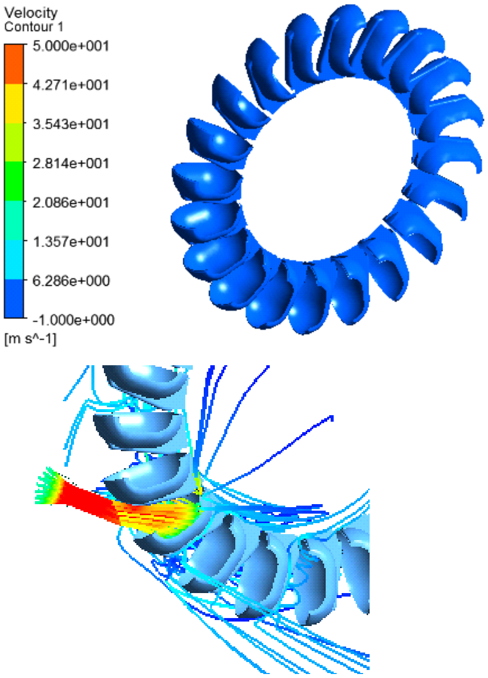
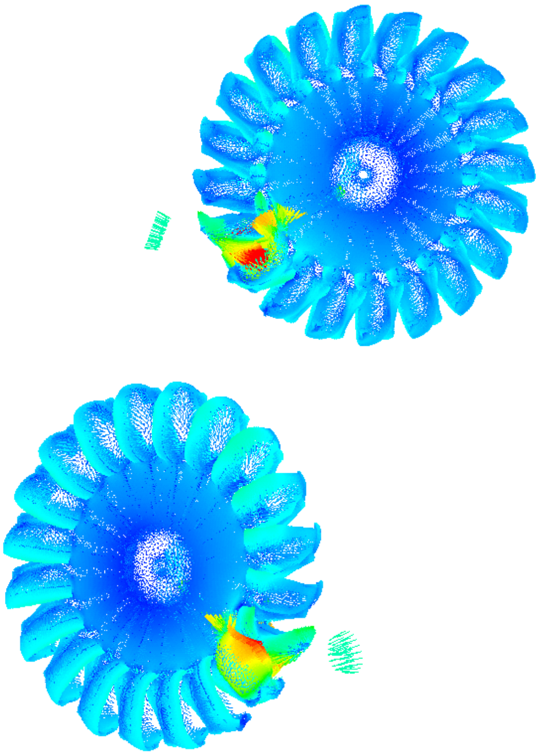


Figure 11. Velocity distribution on the buckets



(a) (b)

Figure. 12 Flow velocity vector on buckets and disc (a) front surface (b) back surface

Figure 12 (a) shows the flow velocity vector on buckets from the front surface and (b) shows the back surface of the buckets according to the flow velocity vector.

IV. RESULTS AND DISCUSSION

From the simulation results, the pressure is maximum at the spot where the jet strikes the bucket, and then decreases towards the edges of the bucket. If the pressure increases, the force will increase proportionally and the bucket can be more failure.

Moreover, from the CFD simulation, the torque (T) of the turbine can be produced and the theoretical torque (T) can be received from the equation $T = Q\ D\ (v_j - U)$. Therefore, the power (P) can be calculated using the formula $P = T\ \omega$. In addition, the CFD simulation results depict the streamlines, water velocity, and water velocity distribution on the bucket. The CFD simulations were validated with the theoretical results. Hence the power and torque are compared at various jet speed. From the theoretical results, the torque is 3.89 Nm if the jet velocity is 3.73 m/s and the torque will become 14.11 Nm when the jet velocity increase to 13.73 m/s. The torque is 24.39 Nm at 23.73 m/s of jet velocity. If the jet velocity increases, the torque of the turbine will increase gradually.

Figure 13. Comparison of torque between theoretical and Numerical results

Figure 13 compares the torque given by the theoretical calculations and CFD. The results show that the value of torque of the turbine varies when the jet velocity changes. If the jet velocity increases, the torque will increase proportionally. And then the power also varies accordingly because the power is directly proportional the torque of the turbine. Figure 14 shows the relationship between the jet velocity and power.

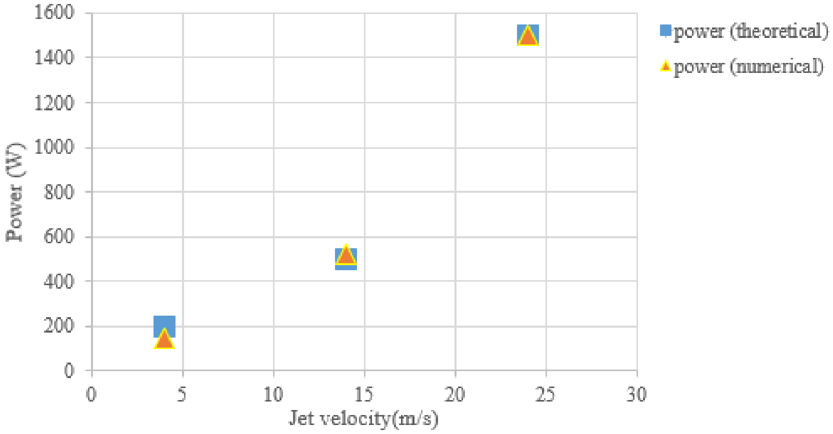


Figure 14. Comparison of power between theoretical and CFD

The power of the Turgo turbine is dependent on the flow rate, head and jet velocity. From figure 14, it can be seen clearly that the power will increase when the jet velocity increases. Therefore the jet velocity is the important point in the designing of Turgo turbine.

V. CONCLUSION

The Turgo turbine is an impulse turbine and it is designated for medium to high head application. But in this research, the head of the turbine is 10 m and the flow rate is 436.9 liter /min. The simulation was performed in symmetrically for 20 buckets and the nozzle angle is 20°. The results obtained from the simulations were dependent on the accuracy in domain set up, quality of mesh, turbulence model and boundary conditions. Therefore, the results can be improved by increasing the mesh quality and the number of mesh nodes.

Moreover, in the designing of the Turgo turbine, the jet ratio and jet velocity play an important role. If the jet velocity increase, the torque of the turbine will also increase. Therefore, the jet velocity is very important in order to get more power. The theoretical power that is developed from the analytical calculation is 520 watts and the jet velocity is 13.70 m/s. Moreover, the simulation shows that the power is 145 watt when the jet velocity is decreased to 3.70 m/s and 1504 watt will be obtained when the jet velocity is increased to 23.70 m/s.

Acknowledgement

The author wishes to express the deepest gratitude to her parents, for their moral support and encouragement. The author is grateful to Dr. Sint Soe, Rector, Mandalay Technological University. Great thanks to Dr. Htay Htay Win, Professor, Head of Mechanical Engineering Department, Mandalay Technological University.

References

1) Barstad, Lorentz Fjellanger,CFD analysis of Pelton turbine, June 2012
2) Anagnostopoulos, J. S., Papantonis, D. E, “Flow Analysis andRunner Design in Pelton Turbines”, 2012.
3) Flow Modeling and Runner Design Optimization in Turgo Water Turbines”, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:1, No:4, 2007.
4) AudriusŽidonis,” Optimisation And Efficiency Improvement Of Pelton Hydro Turbine Using Computational Fluid Dynamics And Experimental Testing”, PhD Thesis, 2015.
5) Gilbert Gilkes& Gordon Ltd. GilkesTurgo Impulse Hydro Turbine, 2016.
6) JonkerKlunne,Wim."Turgoturbine".hydropower.net. hydropower.net. Retrieved 7 March 2016.
7) AudriusŽidonis, George A. Aggidis,” Identifying the Optimum Number of Buckets Using CFD”, Lancaster University Renewable Energy Group and Fluid Machinery Group, Engineering Department, 2012.
8) Bryan R. Cobb, Kendra V. Sharp,” Impulse (Turgo and Pelton) turbine performance characteristics and their impact on pico-hydro installations”, Renewable Energy 50 (2013) pp.959-964, 2013.
9) S. J. Williamson, B. H. Stark and J. D. Booker, “Performance of a low-head pico-hydro Turgo turbine,” Applied Energy, vol. 102, pp. 1114-1126, 2013
10) Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics* (2nd ed.).Pearson Education Limited, Second edition published.
11) W Malalasekera and H K Versteeg (1995), ―An Introduction to Computational Fluid Dynamics‖, the Finite Volume Method, Longman.

12) B. Zoppe, C. Pellone, T. Maitre, P. Leroy (2006), ―Flow Analysis Inside a Pelton Turbine Bucket, Transaction of ASME, Vol. 128, pp.500-511

Authors

First Author – Hnin Hnin Ei, PhD candidate, Mechanical Department, Mandalay Technological University
hninhnin1808@gmail.com
Second Author – Myat Myat Soe, Professor, Mechanical Department, Mandalay Technological University.
myatmyatsoe.mtu@mtu.edu.com