Journal of Physics: Conference Series

PAPER • OPEN ACCESS

Performance evaluation of a bulb turbine designed for ultra-low head applications

To cite this article: Sonal Shandilya *et al* 2019 *J. Phys.: Conf. Ser.* **1276** 012024

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012024 doi:10.1088/1742-6596/1276/1/012024

Performance evaluation of a bulb turbine designed for ultra-low head applications

Sonal Shandilya[1]*, Sujit P. Shelar[1], Shyama Prasad Das[1],Dhiman Chatterjee[1],R. P. Saini[2]
1. Department of Mechanical Engineering, IIT Madras, Chennai 600 036, India.
2. Alternate Hydro Energy Centre, IIT Roorkee, Roorkee 247 667, India.
* Email- me17s042@smail.iitm.ac.in

Abstract

Hydropower is one of the most common forms of clean source of electricity and forms an important source of power for a country like India. This paper is aimed to design mini-hydropower which utilizes ultra-low head and can be used for decentralized power generation and utilization for remote villages. In order to achieve this purpose an axial flow turbine without any guide vanes is designed. This design process is adopted because the presence of inlet guide vanes in the conventional bulb turbines may pose maintenance problem of clogging and also the design is not fish friendly. Another aspect of micro- or mini-hydropower is the non-availability of large reservoirs and there is likelihood that these turbines may be installed as a run-of-the-river type with some seasonal variation of flow rate. The turbine efficiency should be as much as insensitive to the variation in flow rate as possible. In our work, the turbine is expected to produce a relatively flat efficiency flow rate-curve over a range of flow rates. The peak efficiency obtained is about 74%. There is a need to improve the peak efficiency and hence loss analysis is carried out. It was found that the swirling flow in the draft tube downstream of the turbine contributes to a monotonic increase in loss with an increase in flow rate.

1. Introduction

Hydropower potential in India is about 1,45,000 MW of which about 20,000 MW is in the form of small hydropower (with capacity less than 25 MW). A large hydroelectric power plant requires big dams in river which displaces human community, affects the ecosystem and influences free movement of the fishes. It also affects the agriculture and ecosystems in the downstream. Small hydropower, which utilizes flowing water and relies on no or very low head, does not suffer from socio-environmental challenges posed by the construction of big dams. However, of the 20,000 MW, only 4000 MW of this power is currently tapped and there is a target of harnessing 7000 MW at the end of 12th five year plan (2017) in India which itself is much less than the potential. The axial flow hydroturbines are used for such kind of ultra-low head micro/mini hydropower plants applications. This forms the backdrop of the work presented in this paper.

Literature survey suggests that, for ultra-low head applications (heads between 4 and 15 m), bulb turbine can be a very good option [1]. Bulb turbine is an axial flow hydraulic turbine which consists of a bulb-shaped body to accommodate generator, runner and draft tube as part of its geometry. The bulb portion holds a set of inlet guide vanes which direct the incoming flow on the runner blades at the desired angle for which they are designed [2]. The general approach for the design of a conventional bulb turbine, as suggested in available literature, is to start by fixing operating parameters like head (H), flow rate (Q) and arrive at a suitable specific speed (N_S) and then determine different geometric parameters [3][4]. Another method of parametric calculations of the turbine is to fix the outer nominal diameter (D) along with head and flow rate and calculate the specific speed [5][6]. Geometric and flow parameters at hub and tip are calculated using existing correlations from literature [2].Whereas the velocities, flow and blade angles at hub and tip can be calculated by two dimensional cascade theory analysis [6], variation of all these parameters along the blade span are primarily calculated using three dimensional free vortex theory [5-6]. Velocities and their angles inside the turbine are important deciding factors for the geometry of the turbine and the blades. Several algorithms have been developed and tested for different variations of meridional and azimuthal components of velocities along the blade span. Velocities and blade and flow angles are usually taken as linear or higher order function of relative diameter. Apart from runner blades the inlet guide vanes have the considerable effect on turbine efficiency [7]. Once the runner diameter and blade profile are known from the theoretical design, the numerical analysis of the flow inside the turbine passage is performed to save both the money and time before going for manufacturing the desired turbine model. Study has shown that the CFD results predict efficiency and other parameters with good accuracy when compared to experimental test results [8]. A good visualization of flow physics can also be obtained from numerical analysis.

Most of the literature covers the study done on the design of conventional horizontal axis bulb turbine which consists of one set of guide vanes and one set of runner blades. Although the turbine design with inlet guide vanes exhibit the higher efficiency[9-10] the vane-less design is chosen to improve the “off-design performance” of the turbine[11]. Henceforth the present work concentrates on the design of the bulb turbine, without inlet guide vanes, that is capable of producing about 10 kW power under an available head of 4 m and a flow rate 0.27 m^3/s and suitable for run-of-river application.

2. Design Methodology

In order to design the bulb turbine of 10 kW power output, a meanline approach was followed first to design different components and then the numerical modelling of the flow domain was carried out. A full performance map was generated to predict the performance at design and off-design flow conditions. The turbine runner specially designed for this purpose accounted for zero swirl velocity at the runner inlet and a negative swirl component at the outlet to produce required power output. In order to obtain smooth streamlined flow approaching the turbine runner, a contraction cone is used ahead of the turbine runner. The runner section is followed by the conical draft tube.

2.1. Stationary components of the turbine: upstream annular passage and draft tube

The primary purpose of the annular passage is to make the flow approaching the runner section to be uniform. In order to obtain such a streamlined flow the annular area between the bulb and the outer contour is so designed that area decreases smoothly in the forward part and then remains nearly uniform in the rear portion of the passage upstream of the runner. The base design was carried out in an earlier work by us [12]. The bulb shape was made elliptic with the optimum ratio of major axis to minor axis as 2.5.

Downstream of the rotor hub is connected to an ellipsoidal shape nose with major-to-minor axis ratio of 2. This was achieved ensure the flow leaving the runner smoothly, leaving a small wake region.

The shape of draft tube plays an important role especially for high specific speed turbines, since the efficient recovery of kinetic energy at runner outlet depends mainly on it. The inlet of the draft tube is of circular cross section but exit cross section may be either square [13] or circular. In present work, a simple conical draft tube with divergence angle of 10 degree [14] is used to reduce additional losses and manufacturing cost.

2.2. Runner Design

The turbine runner, specially designed for the present work, is accounted for zero swirl velocity at the runner inlet and a negative swirl component at the outlet. To meet the purpose for the design, head, H and maximum flow rate, Q_{max} for the prototype turbine available from the first stage of research, are chosen as the fixed input parameter which are 13.2 m and 1.6 m^3/s respectively, whereas efficiency and specific speed of the prototype turbine were varied in the suitable steps to generate the design space iteratively. The efficiency and specific speed are varied respectively between 0.75 - 0.9 and 600 - 900 for primary calculations.Using these inputs, flow and geometric parameters were evaluated.at hub and tip locations of prototype turbine through the typical graphs generated from experiments conducted on Kaplan turbine [2]. After the calculation of geometrical dimensions, rotational speed, flow parameters and efficiency for the prototype turbine, same are estimated for model turbine by dimensional similitude analysis. Once the hub and the tip diameter for the model turbine are determined, the total blade span is divided into the five cylindrical sections and the velocities and angles are calculated at each section. Among all the design points obtained from the algorithm written for the design procedure, the optimum design point was selected on the basis of combination specific speed, hub–tip ratio, and numbers of blades for the obtained head which provides the maximum efficiency. NACA-4 series aerofoil which exhibited the maximum value of C_L/C_D and better range of lift coefficient C_L with change in angle of attack α was employed for blade design. The design flow chart is shown in figure 1.

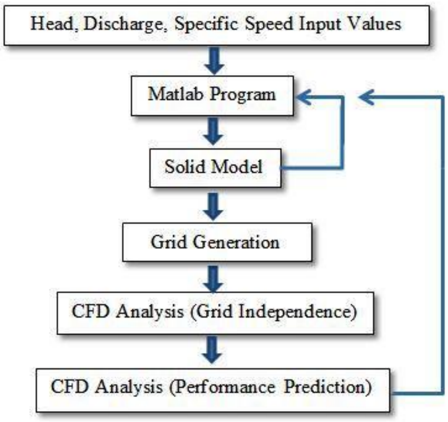

Figure 1 Runner design methodology (Adapted from Ayancik et al.[15])

Overall turbine geometry is shown in figure 2 and dimensions of the major components of the turbine are given in Table 1.

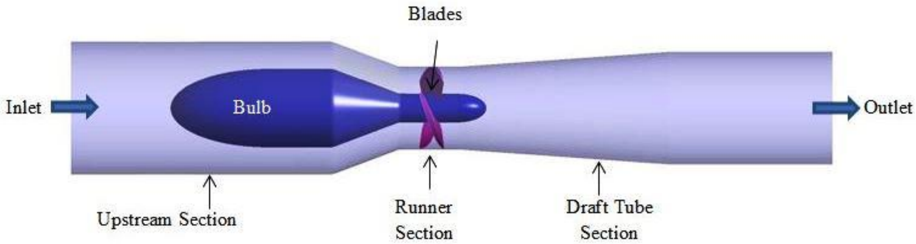

Figure 2 Schematic diagram of the bulb turbine showing the bulb geometry and draft tube

Table 1 Dimensions of major components of the turbine

Upstream	
Casing Diameter (mm)	400
Bulb Diameter (mm)	236
Runner	
Tip Diameter (mm)	250
Hub Diameter (mm)	86
Draft tube	
Half Divergent angle (deg)	5
Outlet Diameter (mm)	340
Inlet Diameter (mm)	250

3. Numerical Simulations

For the numerical investigation of the designed model of bulb turbine, a computational flow domain for the entire bulb turbine model was created which comprised of the upstream section which houses the bulb, runner domain, draft tube domain and one extension part as shown in figure 3. Grid generation for the upstream and draft tube was done in commercial grid generator software ANSYS ICEMCFD. The unstructured hexahedral elements with mesh refinement near the walls are chosen. Grid generation for runner section of turbine was done using the commercial software ANSYS Turbo grid providing structured hexahedral meshes. O grids were generated around the blade surface. The gap configurations due to the runner tip clearance were taken care of by the mesh generator. Typical mesh sizes are 1.1M elements for the upstream section, 1.7 M elements for runner section, 0.7 M elements for draft tube and 0.3 M elements for extension based on grid independence study.

(a) Complete meshed geometry (b) meshed runner section

Figure 3 Computational meshed domain

Although literature suggests that simulation can be performed using k-ε model [16], SST k-ω model is used for the present study. SST (Shear Stress Transport) k-ω is a two equation turbulence model that accounts for transport of the turbulent shear stress and exhibits good behaviour in adverse pressure gradients and seperating flows. Steady state analysis was performed using commercial flow solver ANSYS CFX. High-resolution advection scheme was chosen for these simulations. A frozen rotor mixing model is used at the interfaces between the rotating and non-rotating. Mass flow inlet and average static pressure outlet boundary conditions were selected for these simulations.

Grid independence study was carried out to check sensitivity of the simulation to the number of elements. For this purpose the mesh size factors of 0.5, 0.75.1.0, 1.25 were used. Based on the results of inlet pressure and torque shown in figure 4, obtained mesh of size factor 1.0 is selected for the simulations for performance analysis.

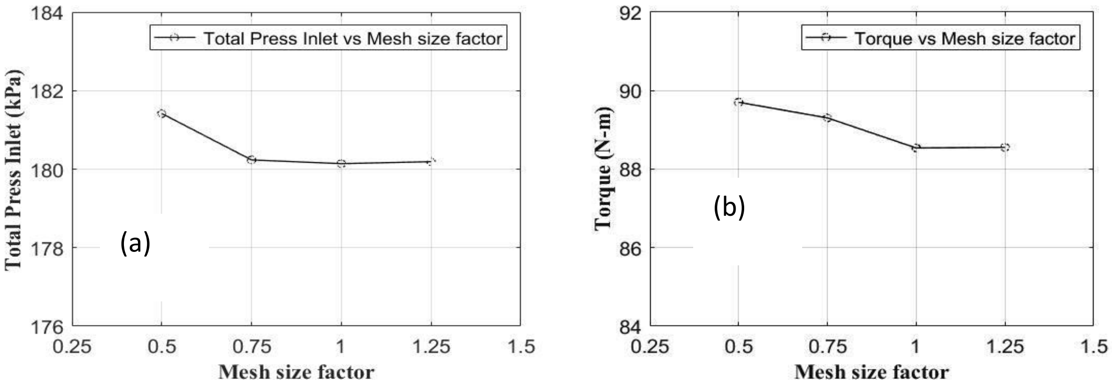

Figure 4 Variation of (a) inlet total pressure and (b) torque with mesh size factor

4. Results and Discussion

In order to predict the performance of the designed bulb turbine, simulations were carried out by varying the flow rate ranging from 25% less to 25% more of the design condition of the turbine. Atmospheric average static pressure at the outlet of draft tube was kept constant in all the cases. The results show that the head and power output obtained from the simulations are 6.74 m and 13.36 kW respectively at the flow rate of 0.274 m^3/s. The efficiency and power output are shown in figure 5.

The hydraulic efficiency of turbine as visible from figure 5(a) is 74 % at the design flow rate and there is little variation over a range of flow rates which is one of the aims of the present study.

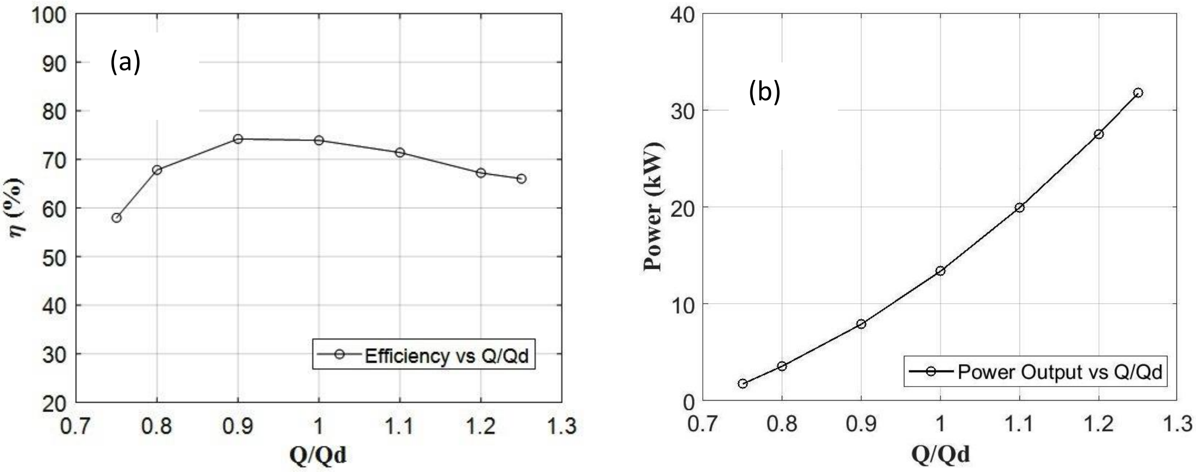

Figure 5 Variation of (a) efficiency and (b) power output with normalized flow rate

The upstream annular passage is intended to provide the uniform flow at stator-runner interface. Figure 6a shows the variation of axial velocity from the hub to the tip of the runner at this location. This variation in axial velocity might be present due to three dimensional flows due to influence of runner. The gap between the inlet and the blade leading edge was kept at 40 mm in order to allow the flow to become more uniform before reaching the leading edge. Figure 6b shows the increase in uniformity of the velocity profile due to the space provided between the leading edge of the blade and the junction between the rotating and stationary portions of the central body.

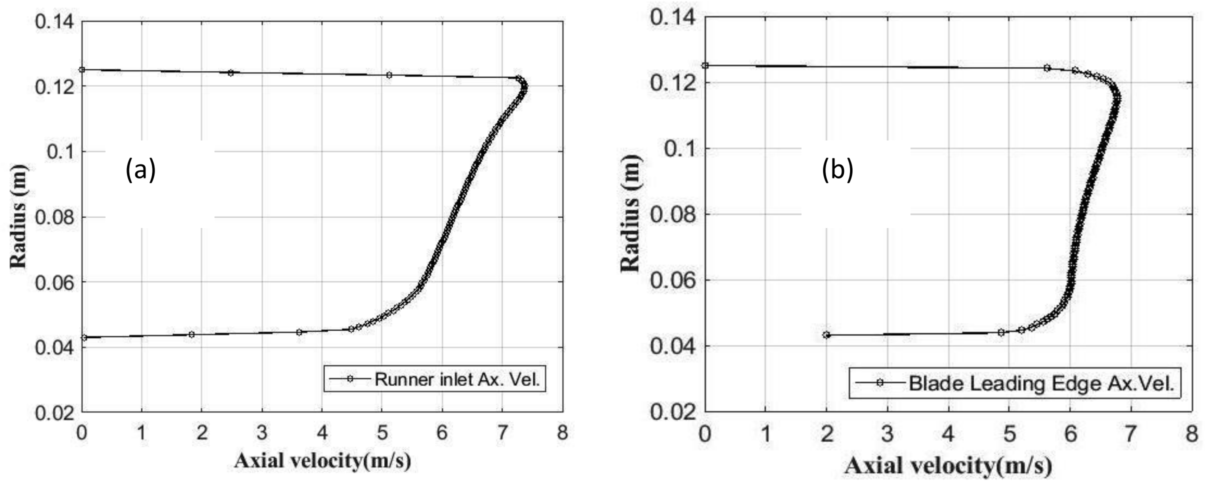

Figure 6 Axial velocity variations from hub to tip of the runner. (a) at the interface between the stator and rotor, and, (b) at the leading edge of the runner blade

Percentage of total pressure loss at different sections of the turbine is shown in figure 7. It is clear that the contribution in percentage loss is the minimum in the upstream section due to optimized geometry of the bulb turbine selected. The loss in the runner section is significantly higher than that in the upstream and particularly at a lower flow rate in comparison to the design condition. This could arise because of several reasons. One of the possible reasons is due to the secondary flow in the blade passage. The tip clearance between the runner and the casing could also give rise to significant losses in this portion of the fluid domain. The flow which leaves the runner section and enters the draft tube section is highly swirling in nature (figure 8) in the draft tube and the loss due to swirling flow increases with an increase in flow rate. Similar argument is valid for the extension portion. Thus, there is an increase in the percentage loss with flow rate for all domains except in the runner.

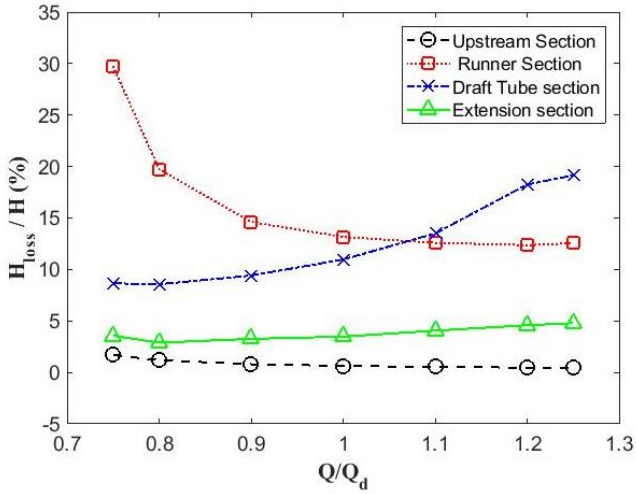

Figure 7 Variation of loss in different domains of turbine with respect to the head utilized by the turbine as a function of normalized flow rate

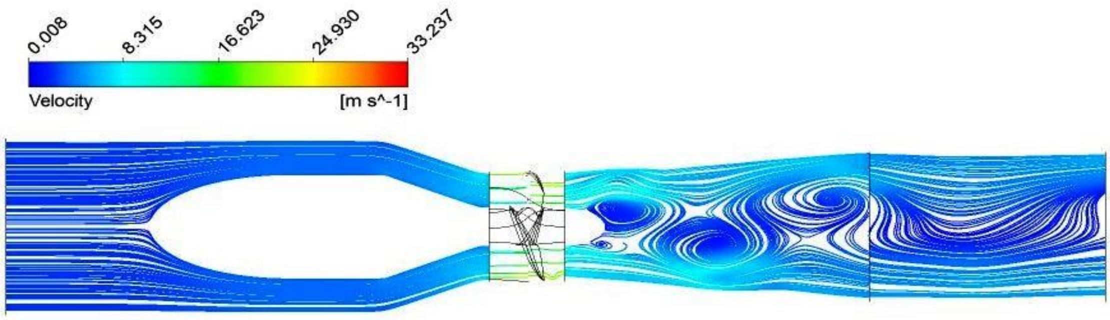

Figure 8 Streamlines in the central plane of the bulb turbine assembly at Q/Q_d=1

5. Conclusion

A bulb turbine, without the conventional guide vanes, has been designed and numerically investigated in the present study. The study shows the potential of achieving desired output from the turbine. Contributions of the losses by different sections are delineated. Hydraulic efficiency of the turbine is slightly lower due to increase in losses at the draft tube section. Further studies are required to minimize the losses in the draft tube section in order to increase the overall efficiency of the turbine.

IOP Conf. Series: Journal of Physics: Conf. Series **1276** (2019) 012024 doi:10.1088/1742-6596/1276/1/012024

References

[1] Bryan Leyland, *Small Hydroelectric Engineering Practice*. CRC Press, 2014.

[2] M. Nechleba, *Hydraulic Turbine: Their Design and Equipments*. 1957.

[3] R. Fraser, C. Deschênes, C. O'Neil, and M. Leclerc, "VLH: Development of a new turbine for Very Low Head sites," no. 157, pp. 1–9, 2007.

[4] Y. Nishi, Y. Kobayashi, T. Inagaki, and N. Kikuchi, "The Design Method of Axial Flow Runners Focusing on Axial Flow Velocity Uniformization and Its Application to an Ultra-Small Axial Flow Hydraulic Turbine," *Int. J. Rotating Mach.*, vol. 2016, 2016.

[5] V. D. Hien, S. Chirarattananon, and K. Luang, "The Development of Bulb Turbine for Low Head Storage Using CFD Simulation," vol. 08, no. 01, pp. 461–475, 2007.

[6] P. Sutikno and I. K. Adam, "Design, Simulation and Experimental of the Very Low Head Turbine with Minimum Pressure and FreeVortex Criterions," *Int. J. Mech. Mechatronics Eng. IJMME-IJENS*, vol. 11, no. 01, pp. 9–16, 2011.

[7] Q. F. Li, H. Quan, R. N. Li, and D. J. Jiang, "Influences of guide vanes airfoil on hydraulic turbine runner performance," *Procedia Eng.*, vol. 28, no. 2011, pp. 703–708, 2012.

[8] V. Bijukchhe, "Comparison of experimental results of horizontal kaplan turbine with computational fluid dynamics," University of Iowa, 2012.

[9] Y. Luo, X. Liu, Z. Wang, Y. Xiao, C. He, and Y. Zhang, "Optimization of the runner for extremely low head bidirectional tidal bulb turbine," *Energies*, vol. 10, no. 6, pp. 1–13, 2017.

[10] S. H. Wang, Z W, Luo, Y Y, Zhang, J, Chen, Q F, Li, X G, Xiao, Y X, Ahn, "Influence of runner clearance on efficiency and cavitation in kaplan turbine," *IOP Conf. Ser. Earth Environ. Sci.*, vol. 163, p. 012068, 2018.

[11] A. T. Simpson, S. W. T. Spence, and J. K. Watterson, "A Comparison of the Flow Structures and Losses Within Vaned and Vaneless Stators for," *J. Turbomach.*, vol. 131, no. July 2009, pp. 1–15, 2009.

[12] S. P. Shelar, "Design and Prediction of Performance of Bulb Turbine Using Numerical Simulation," IIT Madras, 2017.

[13] J. Feng, W. Li, X. Luo, and G. Zhu, "Numerical analysis of transient characteristics of a bulb hydraulic turbine during runaway transient process," *Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng.*, vol. 0, no. 0, pp. 1–11, 2018.

[14] D. S. C. Dejan B. ILIC, Miroslav H. BENISEK, "Experimental investigations of the turbulent swirl," *Therm. Sci.*, vol. 21, pp. 725–736, 2017.

[15] F. Ayancik, U. Aradag, E. Ozkaya, K. Celebioglu, O. Unver, and S. Aradag, "Hydroturbine Runner Design and Manufacturing," *Int. J. Mater. Mech. Manuf.*, vol. 1, no. 2, pp. 162–165, 2013.

[16] T. C. Vu, M. Gauthier, B. Nennemann, H. Wallimann, and C. Deschênes, "CFD analysis of a bulb turbine and validation with measurements from the BulbT project," *IOP Conf. Ser. Earth Environ. Sci.*, vol. 22, 2013.