

Analysis on Cone Casing Design of 5 kW Bulb Turbine

Dr. Nan Aye Myint

Associate Professor, Department of Mechanical Engineering, Technological University
(Toungoo), Myanmar
E-mail: cherrypan082@gmail.com

Received: January 3, 2019; **Accepted:** January 10, 2019; **Published:** January 14, 2019

Abstract: One of the most reliable one where dynamic force of natural water is renewable. Hydro turbine is a rotary engine that extracts energy from a fluid flow by transferring the potential energy to electrical energy. Depending on the head and water flow rate, the variation of velocity, pressure and momentum cause the runner blade to rotate. This paper analyses the various types of cone casing design for bulb turbine which help in improving the turbine efficiency. The velocity and pressure distribution on bulb cone of hydro bulb turbine which consists of runner with four blades and rotating at 1000 rpm is analysed by using SolidWorks Software. At the design head of three meter and flow rate of 0.284 m³/s, the simulation was applied on varying bulb cone shape of quadratic curve, semicircle and conical shape for comparing maximum and minimum inlet and outlet velocity of guide vane between the theory and simulation. From the simulation result, the inlet velocity of guide vane for the cone shape of semi-circle and conical shape is higher than that of three quadratic curves shape. Both types of cone shape produced the inlet velocity of 5.3 m/s and outlet velocity of 7.6 m/s. Among these two types, semicircle cone shape is chosen for this research because it is easier to construct than the conical shape.

Citation: Nan Aye Myint. 2019. Analysis on Cone Casing Design of 5 kW Bulb Turbine. International Journal of Recent Innovations in Academic Research, 3(1): 153-160.

Copyright: Nan Aye Myint., **Copyright©2019.** This is an open access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

1. Introduction

Hydro-electricity power is a well-established technology, which has been producing form power at competitive prices for about a century. Bulb-turbine is a kind of reaction turbine in which the Kaplan runner and the generator are mounted on a horizontal shaft. The ideal head is about 10 meters in height which can produce up to 5 kW of power output. The shaft bearings and the generator are located in a bulb-housing which is supported by piers and which is completely surrounded by water. In the case of a bulb turbine, the bulb portion which is placed in water passage is accompanied by a large displacement toward axial direction during the operation [4]. The axial layout and converging inflow of the bulb turbine enables it to be run at high specific rates of flow, which are conducive to high efficiencies. The straight draft tube of an axial unit is also much more efficient than the draft elbows of conventional turbines. A further attractive feature of bulb units is that they require much less structural work. Furthermore, as most of a bulb unit power house is under water, it only requires very little in the way of superstructure. The guide vanes are either fixed or adjustable, frequently the former [2].

2. Bulb Turbine

Generally, turbine can be classified according to the head and quantity of water available such as high head turbine with low flow rate of water, medium head turbine with medium flow rate of water, low head turbine with high flow rate of water. Since the nature of the river is very difficult to be changed, therefore the Bulb turbine itself needs to do something in order to overcome this situation [1]. Based from Figure 1, Bulb turbine can be divided into few modules which are the moving runner or blade section, the fixed guide vanes, shaft and generator.

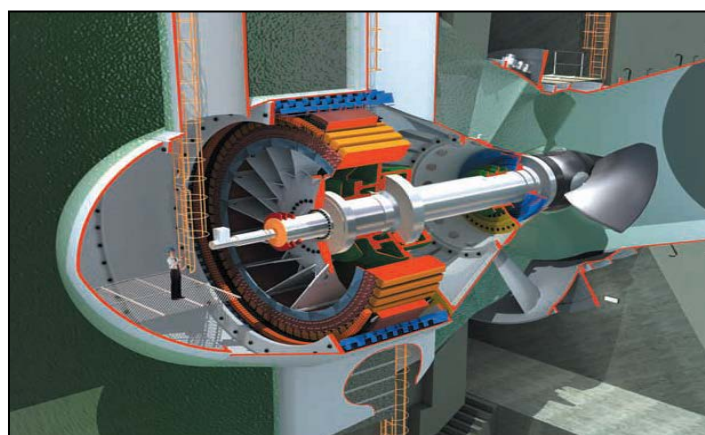


Figure 2.1. Bulb Turbines [4]

In this case, shaft is a component which connects the blade and the generator. It clearly shows that the speed rotation of the blade play very important roles in order to produce more current from the turbine. In this research, the turbine's bulb cone casing will become as a main focus. Five case of cone design will be evaluated in order to improve the performance of blade's speed rotation and increase the power output produce by the Bulb turbine. In the case of Bulb turbine, the size of the bulb cone is depending on the size of the generator. In this paper, to achieve the recommended $n_s < 250$, either the flow is limited or the speed must be reduced. So, it is needed to use 1000 rpm of generator speed as it is suitable for direct drive of a standard 6-pole 50 Hz synchronous generator with 400 mm diameter of stator frame and 5 mm of length [3]. So, the diameter of the bulb cone shape is assumed as 400 mm. With fixed head, and variable parameter of water flow rate, the data collection of power for the turbine will be done.

3. Introduction to Solidworks Flow Simulation

SolidWorks flow simulation is a fluid flow and heat transfer analysis software fully integrated in SolidWorks. SolidWorks flow simulation simulates the model prototype in its working fluid environment. It helps us to answer the question of what are the fluid flow effects on the prototype and the prototype effects on the fluid flow. It is used by students, designers, analysts, engineers, and other professionals to produce highly efficient designs and/or optimize their performance.

Flow Simulation solves the Navier-Stokes equations, which are formulations of mass, momentum and energy conservation laws for fluid flows. The equations are supplemented by fluid state equations defining the nature of the fluid, and by empirical dependencies of fluid density, viscosity and thermal conductivity on temperature. Inelastic non-Newtonian fluids are considered by introducing a dependency of their dynamic viscosity on flow shear rate and temperature, and compressible liquids are considered by introducing a dependency of their

density on pressure. A particular problem is finally specified by the definition of its geometry, boundary and initial conditions [10]. Flow Simulation employs transport equations for the turbulent kinetic energy and its dissipation rate, the so-called k-ε model. Flow Simulation employs one system of equations to describe both laminar and turbulent flows. Moreover, transition from a laminar to turbulent state and vice versa is possible. The flow diagram for the design cycle with SolidWorks Flow Simulation is shown in Figure 2.

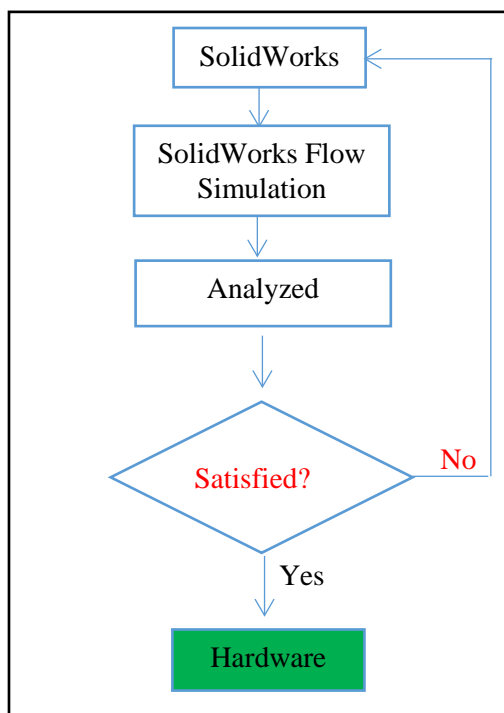


Figure 3.1. Design Cycle with SolidWorks Flow Simulation [7]

3.1 The Finite Volume Method

Analytical solutions are only available for simple problems. They make many assumptions and fail to solve most practical problems. SolidWorks Flow Simulation solves time-dependent Navier-stokes equations with the Finite Volume Method (FVM) on a rectangular (parallel piped) computational mesh. FVM is a general approach for both simple and complex problems. This method is among preferred methods for fluid phenomena modeling [7].

3.2 Computational Domain

Computational domain is a rectangular prism where the calculation is performed. Computational domain's boundary planes are orthogonal to the Cartesian coordinate system's axes. In case of an internal problem, the computational domain envelopes the fluid volume inside a model. If heat transfer in walls is considered, the model walls are also included. In case of an external analysis, the computational domain covers the model's surrounding space [7].

3.3 Numerical Analysis of Bulb Cone Shape

In this thesis, the numerical analysis for the whole assembly that including the outer concrete casing, inner casing or bulb cone, guide vane, runner and draft tube is not sufficient because of the high demand on computer memory and long computation time. So, it is divided into two parts. The first part is the analysis for the assembly of the outer concrete casing, the inner casing or bulb cone and the guide vane. The required parameters are listed in Table 3.1. [7]

Table 3.1. Parameters for First Part of Bulb Turbine

Parameter	Data
Inlet Diameter of Outer Concrete Casing	621.6 mm
Outlet Diameter of Outer Concrete Casing	281 mm
Diameter of Bulb Cone	400 mm
Outer Diameter of Guide Vane	378 mm
Inner Diameter of Guide Vane	280 mm
Number of Guide Vane	10 nos:
Inlet Volume Flow	0.284 m ³ /s

Figure 3.2 shows the assembly of bulb cone, guide vane and outer concrete casing for five cases. Case I, case II and case III have the same cone shape of quadratic curve but different centre radius that is 0.08 m for case I, 0.12 m for case II, 0.16 m for case III. Case IV is semicircle with radius of 0.2 m and case V is conical shape with radius of 0.18 m.

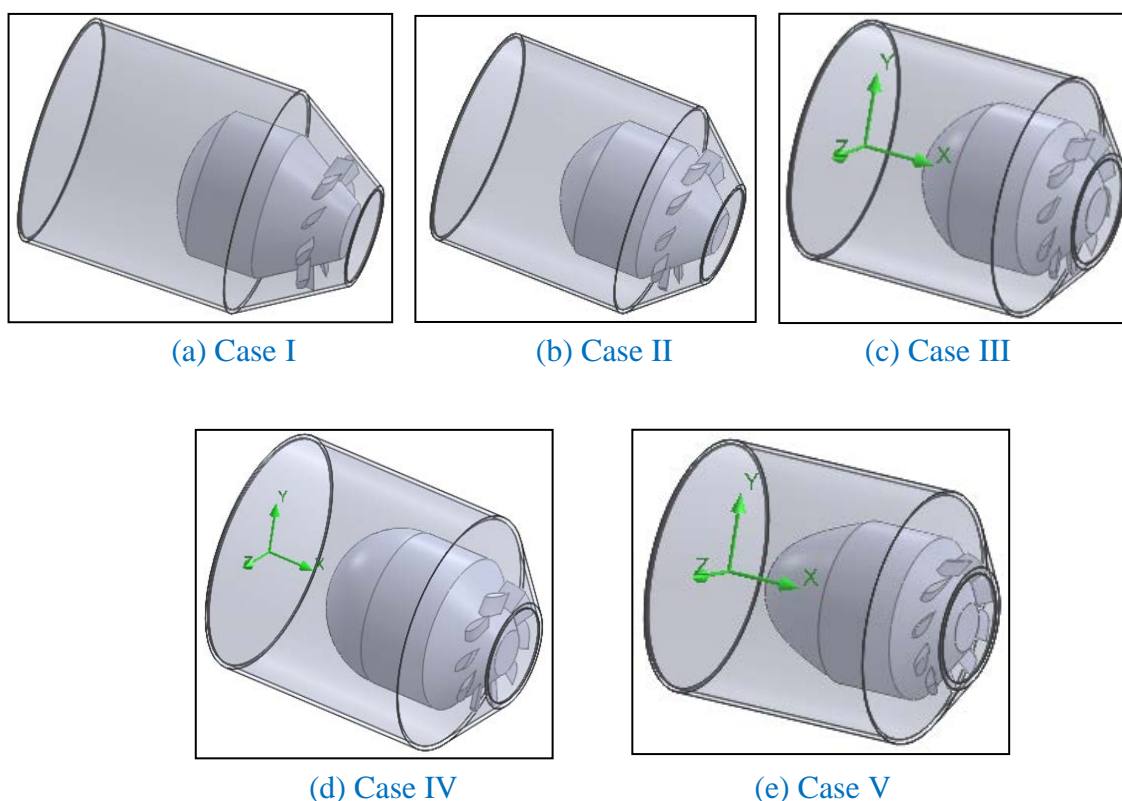


Figure 3.2. Assembly of Outer Concrete Casing, Bulb Cone and Guide Vane: (a, b, c, d and e)

Then, the project for SolidWorks flow simulation is shown in Table 3.2.

Table 3.2 Project For Solidworks Flow Simulation

Project Name	Flow Analysis of Bulb Turbine
Unit System	SI
Analysis Type	Internal
Physical Features	Stationary
Default Fluid	Liquid, Water
Wall Conditions	Adiabatic Wall, Default Smooth Wall
Initial Conditions	Default Conditions

Finally, the inlet and outlet boundary conditions are shown in Table 3.3 and 3.4.

Table 3.3. Inlet Boundary Condition

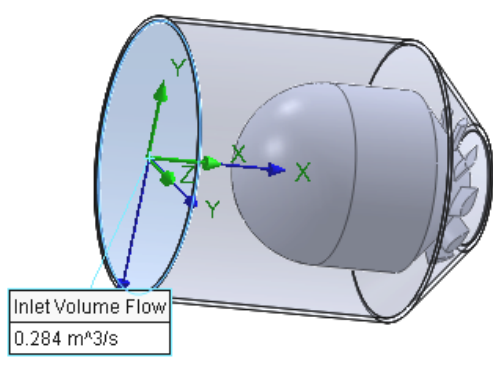
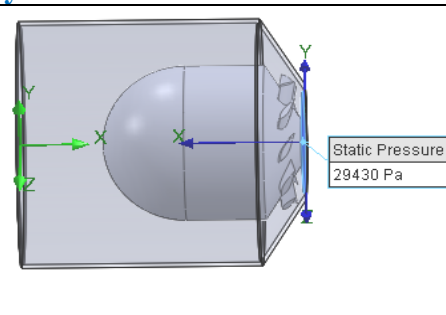
Type	Inlet Volume Flow	
Name	Inlet Volume Flow 1	
Faces to apply	The inner face of the Inlet Lid	
Parameters	Value of 0.284 m ³ /s with Uniform profile in the absolute frame of reference (the Absolute option is selected)	

Table 3.4. Outlet Boundary Condition

Type	Static Pressure	
Name	Static Pressure 1	
Faces to apply	The inner face of the Outlet Lid	
Parameters	Pressure of 29430 Pa with Uniform profile in the absolute frame of reference (the Pressure potential option is disabled)	

4. Results and Discussion

The goal of this simulation is to calculate the flow field parameters such as the pressure and velocity at any point, surface or volume of computational domain.

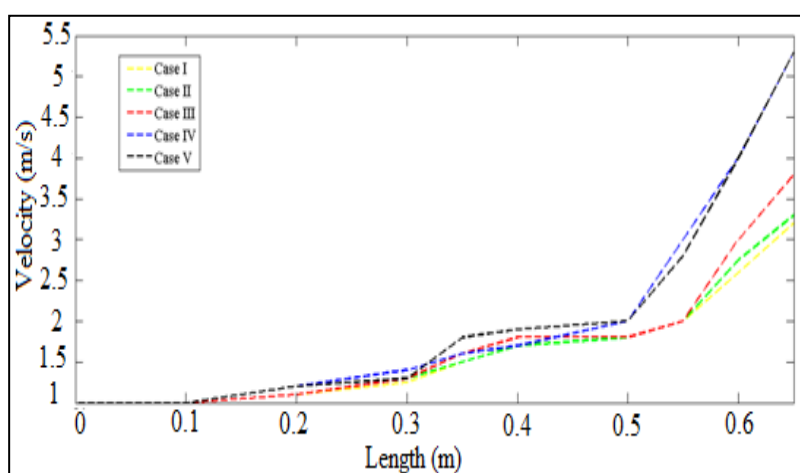


Figure 4.1. Inlet Flow Velocity of Guide Vane

Figure 4.1 shows the flow velocity from the inlet of the concrete casing to the inlet guide vanes. From this figure, the velocity at the inlet of concrete casing for all five cases is nearly the same. But it can be clearly seen that the velocity at the inlet of the guide vane for case IV and case V is 5.3 m/s which is higher than other three cases. In this case, it can be said that the inlet velocity of the guide vane is mainly depending on the shape of the bulb cone. The

more the line of velocity diverge after the stagnation, the less the inlet velocity of the guide vane is. As a result, the cone shape for quadratic curve of case I, II and III cannot produce velocity as high as the cone shape of semicircle and conical shape. Moreover, the outlet velocity of the guide vane for all five cases is also shown in the Figure 4.2.

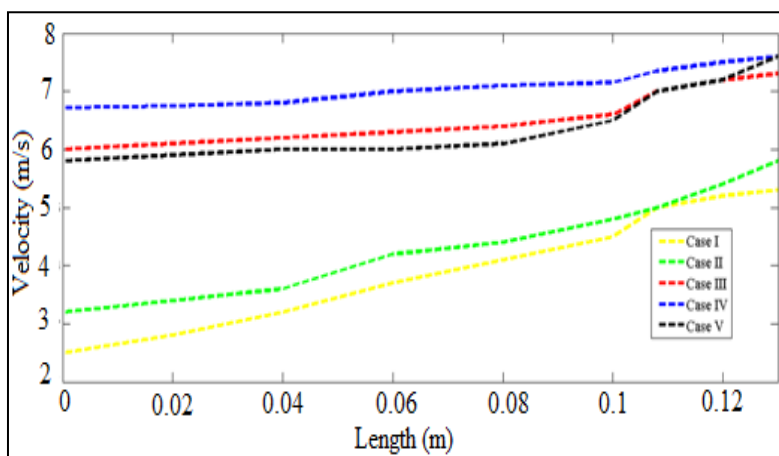


Figure 4.2 Outlet Flow Velocity of Guide Vane

From Figure 4.2, the outlet velocity of guide vane for the cone shape of semicircle and conical shape is also higher than the other three quadratic curves.

Moreover, the comparison of inlet and outlet velocity of guide vane is shown in Table 6.5 and 6.6.

Table 4.1. Comparisons between Theory and Simulation Result of Inlet Velocity of Guide Vane

Type	Parameter	Theoretical Result	Simulation Result	Percent Error
Case I	Inlet velocity of guide vanes (m/s)	4.733	3.1	35 %
Case II			3.3	30 %
Case III			4.8	1 %
Case IV			5.3	10 %
Case V			5.3	10 %

Table 4.2. Comparisons between Theory and Simulation Result of Outlet Velocity of Guide Vane

Type	Parameter	Theoretical Result	Simulation Result	Percent Error
Case I	Outlet velocity of guide vanes (m/s)	5.68	5.3	6 %
Case II			5.8	2 %
Case III			7.2	21 %
Case IV			7.6	25 %
Case V			7.6	25 %

And then, the variation of guide vane inlet and outlet velocity and pressure for five different cone shapes are clearly shown in Figure 4.3 and 4.4.

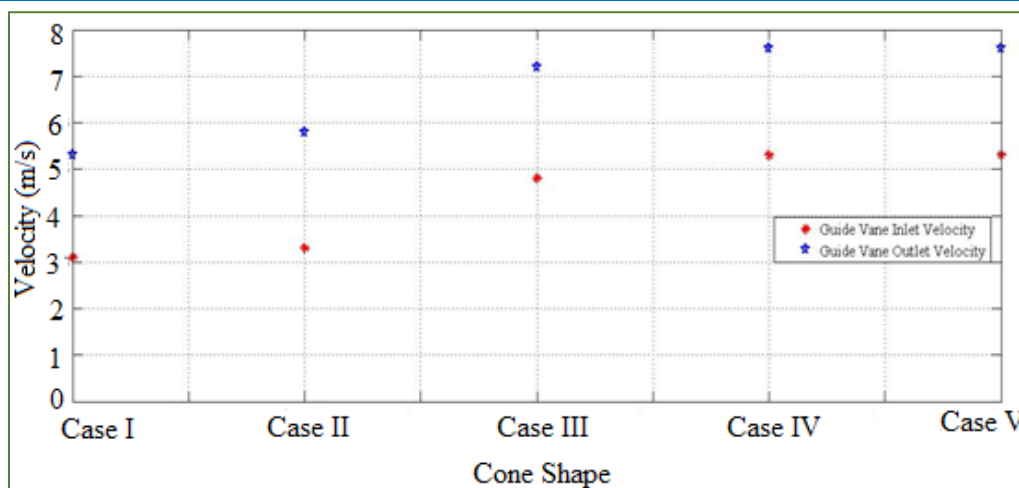


Figure 4.3. Variation of Inlet and Outlet Velocity of Guide Vane for Five Different Cone Shapes

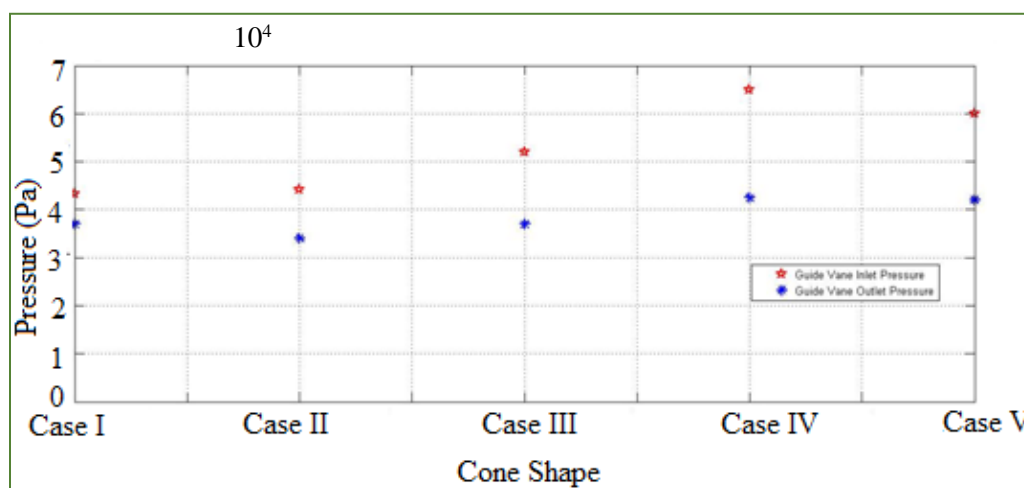


Figure 4.4. Variation of Inlet and Outlet Pressure of Guide Vane for Five Different Cone Shapes

From the simulation result of the five cases, it can be concluded that the case III can be used for this design. However, the case IV and case V are more suitable for this design because the simulation result is more than the theoretical value.

5. Conclusion

The main feature of Bulb turbine compare with other turbine is that there is a divergence inlet flow velocity because of the bulb cone shape. In order to gets the adequate velocity striking the runner blade, the size and shape of the Bulb cone is very important. In this case, the size of the Bulb cone is depending on the size of the generator. The shape of the Bulb cone is considered as quadratic curve, semicircle and conical shape. After that, simulated both types of shape and made a comparison with theoretical result by SolidWorks software.

6. Recommendations

In the case of flow analysis, this analysis method can also be used for other types of turbine. In order to know which type of bulb cone can give the highest inlet velocity of the runner, the type of software used for drawing is needed to consider. It is because there may have some errors when transferring into the analysis software. So, if possible, the next researcher should

check the graphic type when transferring from one type to another included the mesh element size.

References

1. <http://www.hfm.tugraz.at>
2. <http://www.koessler.com>
3. <http://www.picohydro.org.uk>
4. <http://www.alstom.com>
5. Seppo A. Korpela., Principles of Turbomachines
6. Miroslav Nechleba, Dul Techn., M. E., Hydraulic Turbines
7. Anonymous: SolidWorks Flow Simulation Technical Reference, 2nd Joint ASME Fluid Engineering Conference, Canada, (2012).