International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 5, May - 2014

Analyze the Effect of Different Tip Clearance on Performance of Bulb Turbine using ANSYS

Manish S. Maisuria
Mechanical Department,
CGPIT, Bardoli
India

Devendra A. Patel
Mechanical Department,
CGPIT, Bardoli
India

Twinkal Bhavshar
Mechanical Department,
CGPIT, Bardoli
India

***Abstract*— Hydro turbine is a rotary engine that extracts energy from a fluid flow by transferring the potential energy to electricity generation. Depending on head and water flow rate, variation of pressure and momentum cause the runner blades to rotate.**

This research studied the effects of clearance between casing and turbine runner blade which help in improving turbine efficiency. To take varies clearance value and see the effect on bulb turbine performance to help ANSYS software. The bulb turbine consists three runner blade, 1685 runner speed, guide vane angle 35^{0} and 18 no of guide vanes. The simulation was applied on varying clearance for comparing the maximum and minimum pressure on blades.

Keywords—bulb turbine, clearance, efficiency, Ansys

I. INTRODUCTION

As the result of dramatic rise of fuel cost, the development of extreme low head hydro potential by bulb turbines. Bulb turbine is applying for low head application. In India where there have been persistent shortages of power supply in the recent years, hydraulic power being developed as well as thermal and nuclear powers in high pace as a countermeasure of the solution of this problem.

Computational Fluid Dynamics (CFD) is a useful tool to optimize the design of hydro turbines and improve in their efficiencies; CFD is used to analyze the fluid flow helping the design part to be cost and time saving. The CFD simulation of fluid flow pass hydro turbine is to analyze the effect of blade angle, inlet guide vanes and runner blades, on pressure and velocity distributions of hydro bulb turbine. And the results would be useful as the guideline for blade design of hydro bulb turbine.

II. IMPORTANCE OF CFD

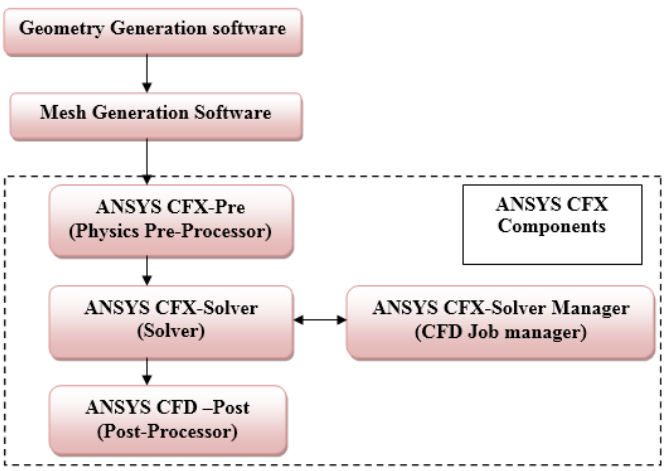
CFD is one of the most important tools available in our world. It is a virtual modelling technique with powerful visualization capabilities and one can evaluate the performance of a wide range of system configurations on a computer. It is cheaper to create a model of the underlying physical processes and test alternative configurations than to build a real prototype and have to change it later based on trial and error. By simulation a model can be tested with causing risk to human life. Testing using simulation also saves lot of time. E.g. Building an aircraft requires hundreds of millions of rupees and couple of years. But by using simulation, a number of models of aircraft can be tested without risking life of a pilot, in fairly less time and minimal cost."

III. COMPUTATIONAL FLUID DYNAMICS MODEL

CFD is software which is designed to deal with the entire problem related to fluid dynamics. Thus, ANSYS incorporated this software in it as a module. ANSYS also designed its own module for fluid dynamic problems known as "CFX". It can be used in standalone mode and also can be used from the ANSYS WORKBENCH. Particularly it has following three main components in standalone mode:

1. CFX Pre
2. CFX Solver Manager
3. CFX Post

ANSYS CFX consists of four software modules that take a geometry and mesh and pass the information required to perform a CFD analysis.



Specification of bulb turbine

No. of blades	=3
Runner Diameter	=155 mm
Hub Diameter	=55 mm
No. of guide vane	=18
Head	=2 m
Flow rate	=0.069 m3/s

Runner speed =1685 RPM
Guide vane angle =350

After the geometric modelling of the bulb turbine with the help of Creo. It is imported into the ANSYS for further process. The geometry made in the Creo elements is converted in to STEP file (.stp format) and then it is called in the ANSYS for the meshing and the boundary conditions.

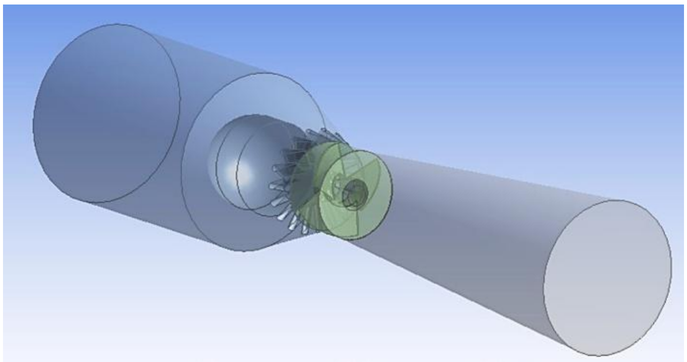


Fig. 1 geometry of bulb turbine

Meshing is done after the geometry is successfully imported. Selecting CFX type meshing for that manually input sizing of meshing cell. In this command the ANSYS generates the meshing automatically according to the sizing data given. The meshing tool in the Workbench along with the 4409968 nodes and 2221070 elements in meshing and fig 2 shows the meshed model of the bulb turbine.

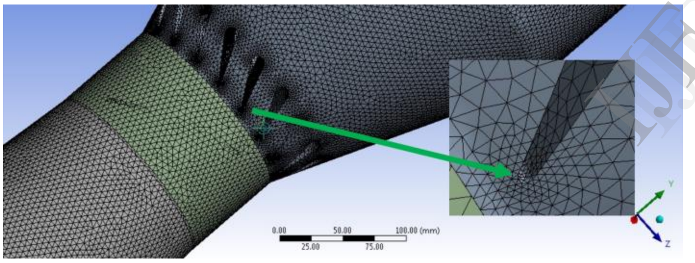


Fig 2 Meshing model of blub turbine

The inlet boundary condition was set by static pressure (N/m^2) and the outlet condition of upstream boundary was set by using mass flow rate (kg/s). Runner speed was set by angular velocity (RPM). This simulation is set to be steady flow study and k-ε was selected for turbulence model.

IV. RESULT AND DISCUSSION

The present research work begins with mesh generation and then effect of clearance on turbine performance is optimized.

Power (P)

$$P = \frac{2\,\pi\,N\,T}{60}$$

Turbine efficiency (η)

$$\eta = \frac{P}{\rho\;g\;Q\;H}$$

Where, N= angular Speed (RPM)

T= rotor torque (N/m2)

ρ= density of fluid (kg/m3)

Q= flow rate (m3/s)

H= head (m)

Simulation work begins with Mesh generation. Here compare to different mesh size with efficiency. When the mesh cell size is low, efficiency of turbine is also low. Gradually increase the mesh size it's shown the efficiency of turbine is increase up to one stage. After that further increase the size of mesh but no effect on efficiency. It's become constant shown in figure 3. So, consider the constant value of mesh size.

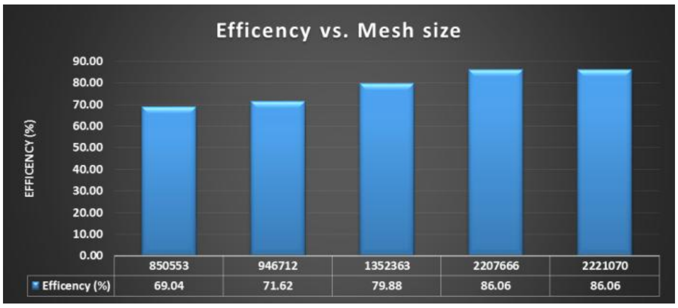


Fig.3 Grid independent test

In order to investigate the effect of mesh size, result of the implemented mesh (2221070 cells) and coarsest tested mesh size (220766 cells) are presented in figure 3. Both meshes predict the measured performance with good accuracy.

V. EFFECT ON BULB TURBINE WITH TIP CLEARANCE OF RUNNER BLADE

On varying distance between the runner tip and casing, there is some effect on turbine efficiency. Distance variation from 0.25 to 1.25 mm with the interval of the 0.25 mm is chosen. Because of bypass or sleep at the tip clearance of water, some of the kinetic energy does not convert rotary or mechanical energy. Therefore some loss creates by the clearance. It is not possible to removing clearance but possible to maintain the minimum the clearance distance. So, improvement of the turbine efficiency occured. Here the effect on and turbine efficiency has shown in figure 4.

Table 1 Effect on power and efficiency on various Tip clearance

clearance (mm)	**Speed (RPM)**	**Mass Flow (Kg/s)**	**Torque (Nm)**	**Input power (kW)**	**Output Power (kW)**	**Effi. (%)**
0.25	1685	69	7.17	1.354	1.265	93.45
0.50	1685	69	7.03	1.354	1.239	91.53
0.75	1685	69	6.87	1.354	1.212	89.52
1.00	1685	69	6.61	1.354	1.166	86.06
1.25	1685	69	6.42	1.354	1.132	83.58

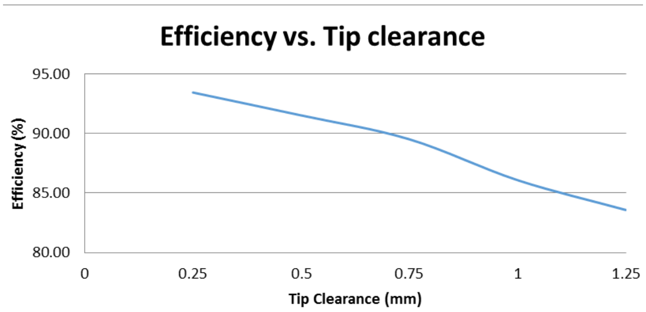


Fig 4 Efficiency vs. Tip clearance

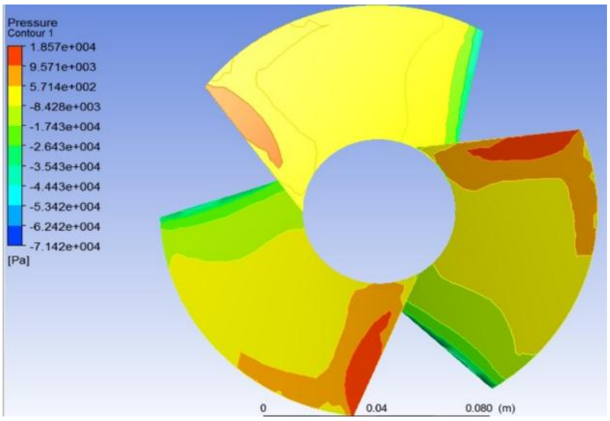


Fig. 5 Pressure Distribution on blade at Pressure side

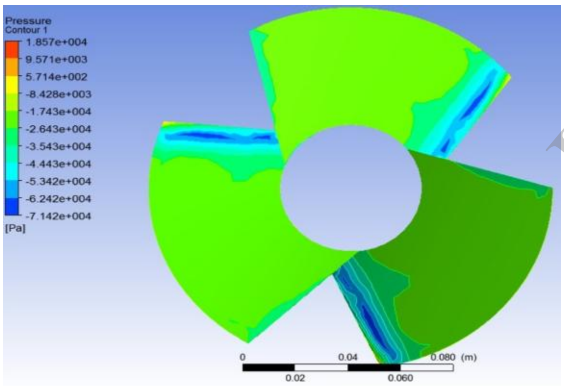


Fig 6 Pressure Distribution on blade at Suction side

Table 2 static pressure on blade

clearance (mm)	Min Pressure (Pa)	Max. Pressure (Pa)
0.25	-72742.2	19643.7
0.5	-73644.1	19613.3
0.75	-71422.9	18570.0
1	-70864.8	19639.9
1.25	-68254.2	18926.4

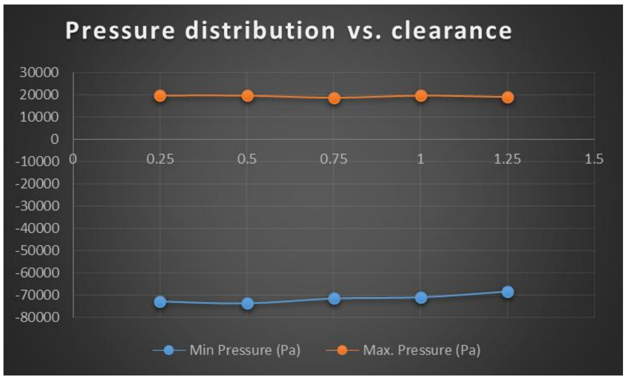


Fig 7 pressure distribution vs. clearance

VI. CONCLUSION

The CFD simulation results in this paper show pressure values on runner blades at specific head and fluid flow.

- The simulation was investigate on varying clearance at different value. The maximum pressure different create for clearance value 0.25.
- Fig.4 shows that clearance between casing and runner blade tip is inversely proportional to efficiency.

By adjusting the clearance gap, it clearly affects the efficiency of the hydro turbine.

REFERENCES

[1] Helmut Benigni, Helmut Jaberg, “Stationary and Transient Numerical Simulation of a Bulb Turbine”; IASME / WSEAS International Conference on Fluid Mechanics and Aerodynamics, Athens, Greece, August 25-27, 2007

[2] Chung, T. J. (2002), “Computational fluid dynamics”; Cambridge University press.

[3] HAN Fengqin, YANG Lijing , YAN Shijie, and KUBOAT Takashi, “New Bulb Turbine with Counter rotating Tandem runner”; DOI: 10.3901/CJME.2012.

[4] Yodchai Tiaple,and Udomkiat Nontakaew,” The Development of Bulb Turbine for Low Head Storage Using CFD Simulation”

[5] Anderson, J. D.(1992) “Introduction to computational fluid dynamics”, Edited by Wendt, John F, New York, Spriger-Verlag.