EFFECT OF RUNNER SOLIDITY ON LOSSES IN MIXED FLOW HYDRAULIC TURBINE

Dr. Ruchi Khare
Assistant Professor, Department of Civil Engineering,
M.A. National Institute of Technology, Bhopal

ABSTRACT

The flow through the turbine components is associated with energy losses and hence energy output from turbine is always less than the input energy. The global performance of turbine is characterized by its overall efficiency. The casing has spiral shape and hence has swirling flow and flow is subjected to frictional and vortex losses. The passage in stay ring and distributor has solid vanes and flow passes through the ducts between the vanes is subjected mostly frictional losses. Since runner is the most important component of mixed flow turbine and its design i.e. the blade profile, number of blades plays vital role in overall performance of turbine at all operating regimes. In the present work the numerical simulation of hydraulic Francis turbine is carried out by using ANSYS software. The losses in each part of turbine are computed at different operating conditions by changing the solidity of existing runner. The numerical results for efficiency of existing runner are validated with experimental values. This study can be used efficiently as a reliable tool for the practical design and performance analysis of Francis hydraulic turbines.

Keywords: Turbine Losses, Francis Turbine, CFD, Draft Tube Loss, Runner Losse.

INTRODUCTION

Modern Francis turbines have developed into very different form from the initial design but they all are based on the initial design of inward flow turbine. It is generally used in medium head power plants. The Francis turbine uses pressure energy as well as kinetic energy of water to produce the power output. The water from reservoir flows through penstock, spiral casing, stay ring, distributor, runner and draft tube. The flow through series of stationary and rotating blade rows and diffuser becomes so complex that theoretical analysis is not possible [Sick, 2008]. The water from distributor enters to rotating runner where frictional, disk friction and shock losses may occur. The flow in draft tube is retarding and meets friction and secondly flow (eddy) losses.
The nature and magnitude of loss in individual component vary depending on flow pattern. Hence it is necessary to assess the losses in individual component separately for improvement of turbine efficiency [Ayancik, 2013]. The losses are either to be computed from theoretically or by experimental model investigations. The proper investigation of losses in turbine may also help to operate the turbine at the most efficient regime.

The numerical simulation using CFD has proven an effective tool for studying detailed flow behavior in mixed flow Francis turbine.

The overall performance of turbine depends on the individual performances of each component. The change in guide vane opening leads to variation in discharge and flow direction, which in turn leads to the variation of flow parameters in different components of the turbine [Helmut, 2008]. Similarly, variation in rotational speed of runner also affects the flow characteristics, especially, in the runner. The runner solidity i.e. the ratio of chord to pitch is again important parameter in runner design as it also affects the flow parameters and also performance of turbine.

As the energy transfer takes place in the runner, its design is of utmost importance. The proper design of blade profile and solidity of runner can optimize the velocity distribution and cavitation which in turn affect the overall performance of turbine [Balint, 2012]. In this paper, the performance of existing turbine has been studied by changing the solidity of the runner. The effect of solidity change on flow parameters like pressure and velocity distribution in the annular space of runner, energy losses in different parts i.e. runner, draft tube is studied and discussed in detail.

**METHODOLOGY**

The 3D geometry of turbine is prepared by using ANSYS Workbench. In mixed flow turbine, runner is rotating and casing, stay vane, guide vane and draft tube are stationary. Hence separate domains namely casing, stay ring, distributor, runner and draft tube are modeled and then assembled through interfaces for simulation.

The existing turbine consists of 18 stay vanes, 18 guide vanes, 13 runner blades and a draft tube. The diameter of runner is 1010 mm. The aerofoil profile of blade at 10 sections from hub to tip are generated by sectional 3D coordinates along its length and to model this profile, it requires generation of sectional 3D co-ordinates along its length. The existing runner has 13 blades and two more runner variations have been modeled with 11 and 15 number of blades to get variation in solidity. The solidities for all three runners (with 11, 13 and 15 blades) at 10 sections between hub to tip are mentioned in Table 1.

<table>
<thead>
<tr>
<th>No. of blades</th>
<th>Section-wise solidity</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>I</td>
</tr>
<tr>
<td>11</td>
<td>2.46</td>
</tr>
<tr>
<td>13</td>
<td>2.92</td>
</tr>
<tr>
<td>15</td>
<td>3.367</td>
</tr>
</tbody>
</table>
The flow domain of turbine is discretized into small tetrahedral elements. The elements are connected to each other at common points known as nodes. The group of elements and nodes in flow domain is known as grid/mesh. Part meshing is applied to different parts of turbine for getting better accuracy. The mesh size is fixed after mesh dependency test and $Y^+$ value required. There are 81834 nodes and 458580 elements at 11 blade runner mesh, 129905 nodes and 671901 elements at 13 blade runner mesh, 422112 nodes and 2346807 elements at 15 blade runner mesh.

**Boundary Conditions**

The flow behavior inside turbine space depends on nature of boundary and values of input parameters at boundaries. The flow simulation was performed at 80.93 mm GV opening, which is the best operating regime of existing turbine. The mass flow rate as 7200 Kg/s is specified at casing inlet as inlet boundary condition and outlet boundary condition is specified at draft tube outlet as relative static pressure equal to 0 Pa. The SST $\kappa$-$\omega$ turbulence model has been used for the analysis due to curvatures in flow path and runner rotation. The walls of all domains are considered as smooth with no slip. The runner domain is rotating and all other domains are set stationary. The analysis is done at 6 different speeds varying from 400 rpm to 900 rpm at an interval of 100 rpm.
Formulae used

The numerical simulation of Francis turbine can take into account only hydraulic losses since it is difficult to simulate other losses in the flow passage. The losses are calculated as

\[
\text{Head loss at runner (\%)} \quad h_{lr} = \frac{TP_s - TP_r}{\gamma H} \times 100 \quad \text{------------------- (I)}
\]

\[
\text{Head loss at draft tube (\%)} \quad h_{ld} = \frac{TP_d - TP_{de}}{\gamma H} \times 100 \quad \text{------------------- (II)}
\]

\[
\text{Speed factor} \quad n_{11} = \frac{n D}{\sqrt{B H}} \quad \text{------------------- (III)}
\]

RESULTS AND DISCUSSIONS

The accuracy of numerical simulation depends on many factors and it is required to carry grid dependency test and validate the results with experimental values. The flow simulation for the best operating regime i.e. 80.93 mm GVO and 600 rpm is carried out by taking three grid sizes. The comparison of normalized turbine efficiency (ratio of actual efficiency to maximum model efficiency) from model test is compared with the efficiency obtained from CFD results are shown in fig. 3 at some operating points. The efficiency values and its variation obtained in both the cases bears close comparison and hence validate the CFD results.

The analysis was performed at constant guide vane opening at six speeds varying from 400 rpm to 900 rpm at an interval of 100 rpm. It is observed that there is no significant effect of change in solidity of runner in the losses at casing, stay ring and distributor for all operating conditions since the flow path till the runner is not changed for all the three cases of solidity. The effect of solidity is observed only in runner and draft tube performance is being affected [Tridon, 2010] [Enomoto, 2012]. The percentage loss in runner is plotted against speed factor In fig.4. The variation of losses in runner at three solidities indicates that minimum loss occurs near designed speed due to minimum shock losses. The point of minimum loss shifts towards lower speed as the solidity increases. The runner loss increases with either increase or decrease in speed from designed speed i.e. 600 rpm.

It is seen from fig.5 that the draft tube loss slightly decreases with increase in solidity at all speeds. It also has minimum value around designed speed factor and it may be due to changes in whirl going out from runner at different speeds.
It is observed that the losses in runner and draft tube significantly affect overall efficiency of the turbine (fig.6). The hydraulic efficiency initially increases with increase in speed and attains maximum value near designed speed and then starts decreasing with further increase in speed at all solidities. The similar variation in efficiency at fixed solidity is also obtained in experimental testing of turbine models. As the solidity increases, the maximum efficiency point shifts towards lower speed value. The best efficiency is obtained for runner with 13 blades.

It is seen from fig.7 that the flow pattern at runner entry is nearly same for all solidities. The pressure at runner inlet increases slightly with increase in solidity. It is due to change in flow area in runner space. There is decrease of pressure from inlet to outlet of runner which is the characteristic of reaction turbine. The streamline pattern in draft tube for three solidities shown in fig. 8 depicts that whirl coming out of runner decreases with increase in solidity.
CONCLUSION

- It is seen that the meridional and whirl components of velocity coming out of runner are dependent on the operating regime of turbine.
- The losses in casing, stay ring and distributor remain nearly unaffected due to change in solidity but losses in runner and draft tube has been found to be affected by solidity.
- The variations of losses in runner and draft tube have parabolic pattern with speed. The minimum loss occurs close to rated speed.
- It is found that the point of maximum efficiency or minimum loss shifts towards higher speed factor with decrease in solidity.
- The loss characteristics at runner and draft tube using CFD will be useful for design optimization of runner give the best performance.

Nomenclature

- \( D \) - diameter of runner (m)
- \( G \) - acceleration due to gravity
- \( h_{lr} \) - head loss in runner (%)
- \( h_{ld} \) - head loss in runner (%)
- \( H \) - net head (m)
- \( n_{11} \) - unit speed
- \( n \) - speed of runner (rpm)
- \( TP_{ri} \) - total Pressure at inlet of runner (pa)
- \( TP_{ro} \) - total Pressure at inlet of runner (pa)
- \( TP_{di} \) - total Pressure at inlet of runner (pa)
- \( TP_{do} \) - total Pressure at inlet of runner (pa)
- \( \gamma \) - specific weight of water (N/m³)
REFERENCES

[1] Lewis R.I. Developments of actuator disc theory for compressible flow through turbo-

computational fluid dynamics applications", Proc Instrn Mech Engrs Vol 213 Part C, 1999,
85-102

surrogate models, Ph D thesis, Department of Applied Physics and Mechanical Engineering,
Lulea University of Technology, 2006.

original and redesigned hydraulic turbine draft tube, Journal of Advances in Engineering


Analysis of the Swirling Flow in a Francis Turbine Draft Tube: Focus on Radial Velocity

Optimization of Runner Blades for Reaction Hydraulic Turbines”, 26th IAHR Symposium on

Speed Francis Turbine Runner”, 26th IAHR Symposium on Hydraulic Machinery and

and Manufacturing”, International Journal of Materials, Mechanics and Manufacturing,
Vol 1, No 2, 162-165.

Performance Characteristics of Elbow Draft Tube with Dividing Pier”, International Journal
of Mechanical Engineering & Technology (IJMET), Volume 5, Issue 5, 2014, pp. 1 - 12,
ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.

Beds”, International Journal of Civil Engineering & Technology (IJCIET), Volume 5,

Ventilation for Hot Climate by using CFD”, International Journal of Civil Engineering &
Technology (IJCIET), Volume 5, Issue 2, 2014, pp. 71 - 80, ISSN Print: 0976 – 6308,
ISSN Online: 0976 – 6316.

Characteristics Over Plane and Serrated Slopes”, International Journal of Civil Engineering &
Technology (IJCIET), Volume 4, Issue 4, 2013, pp. 185 - 193, ISSN Print: 0976 – 6308,
ISSN Online: 0976 – 6316.