DEVELOPMENT OF CROSS FLOW TURBINE
FOR PICO HYDRO

R.P. Saini and S.K. Singal
Alternate Hydro Energy Centre, IIT Roorkee, Uttrakhand, 247667
Email: saini.rajeshwer@gmail.com

ABSTRACT

Lot of potential of hydro power in the range of micro hydro exists in hills in India. Cross-Flow turbine has been considered simple and less costlier turbine for such sites. This turbine basically is an impulse type turbine suitable for medium/low head conditions and generally recommended for smaller capacity. It is very simple in construction and can be fabricated locally. However, the efficiency of cross flow turbine is considered to be poor and can be enhanced by doing suitable design. Under the present paper, a CFD based design approach of a 5.0 kW capacity cross flow turbine is discussed and presented in order to develop the optimal design under different operating conditions.

Keywords: Cross flow turbines, CFD, Design, Hyperworks-12.0

1. INTRODUCTION

In hilly region micro hydro up to 100 kW have momentous role in utilization of mechanical power and electricity generation. The capacity of micro hydro power plant up to 5.0 kW is considered under development of water mill program by Ministry of New and Renewable Energy (MNRE), Govt. of India. The popularity of the turbines under micro hydro lies in the fact that they are less costly and can be fabricated locally. There are various types of turbines that can be used in micro hydro. Among them, cross-Flow turbine has been considered technoeconomically viable for such sites. Cross flow turbine runner can be fabricated locally, but has the poor efficiency. Khosrowpanah [1] carried out a study of cross flow turbine experimentally by varying the number of blades. Desai and V. Rangappayya [2] carried a parametric study of cross flow turbine analysis to find out the key parameters influencing the turbine efficiency. Effect of draft tube size on the performance of a cross-flow turbine was studied by H. Reddy et al. [3]. Joshi [4] conducted test on a 5 kW cross flow turbine at IIT Delhi, India and demonstrated that the addition of the draft tube increases efficiency at lower heads but that the efficiency deteriorated at higher heads. Nozzle flow in a cross flow turbine was studied by N. H. Costa Pereira and J. E. Borges [5]. Investigation of the performance of a cross flow turbine was made by Hayati Olgun [6]. An experimental investigation was conducted to study the effects of some geometric parameters of runners and nozzles (e.g., diameter ratio and throat width ratio) on the efficiency in the cross-flow turbines, by varying of ratio of inner-to-outer diameters of runners and gate openings of two different turbine nozzles under different heads. Under this paper an attempt has been made to carry out a CFD based design of cross flow turbine.
2. WORKING PRINCIPLE OF CROSS FLOW TURBINE

Cross flow turbine works in two stages where water enters through the nozzle and strikes with the blade surface at first stage and after crossing the open space inside the runner then strike the blades again at second stage and then discharge through outlet. The water crosses the shaft before leaving the turbine hence the name is given as ‘cross flow’. Fig.1 depicts the schematic of water flow over the turbine runner.

![Schematic of cross flow turbine showing the flow over the runner blades](image)

Fig.1: Schematic of cross flow turbine showing the flow over the runner blades [7]

In cross flow turbine runner and nozzle are main parts. The nozzle guides and controls the water flow into the runner and converts the potential energy into kinetic energy in the form of high velocity jet. Nozzle is rectangle in cross section. The two surfaces are plane and other two surfaces are typically curved. Other important consideration is of runner design for CFT is selecting the optimum number of blades. If the blades are too few as well as excessive number of blades then both of conditions may increase hydraulic losses and reduce efficiency. The runner blades can be cut from a standard sheet metal or steel pipe and then be bent into the required blade profile. In some cases, to improve on the structural integrity of the runner, more than two equally spaced discs are employed.

3.0 DEVELOPMENT OF CFD BASED DESIGN OF CROSSFLOW TURBINE

In order to discuss the CFD based design of a cross flow turbine, a capacity of 5.0 kW for turbine for the site parameters given in Table 1 is considered. The 3D model of cross flow turbine is done in 3D CAD software Pro-e. The method to design the components is discussed below.

<table>
<thead>
<tr>
<th>Table 1: Parameters considered for design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power (P)</td>
</tr>
<tr>
<td>Head (H)</td>
</tr>
<tr>
<td>Discharge (Q)</td>
</tr>
<tr>
<td>R.P.M. (N)</td>
</tr>
<tr>
<td>Efficiency (η)</td>
</tr>
</tbody>
</table>

260
3.1 Design of Blade Profile

There are various design factors or parameters that affect the turbine performance such as outer diameter of runner ($D_1$), inner to outer diameter ratio, attack angle ($\alpha$), optimum number of blades. Values of different design parameters considered are given below.

- Diameter ratio ($D_2/D_1$) : 0.666 (standard value for CFT)
- Outer Diameter of runner ($D_1$) : 300mm
- Inner Diameter of runner ($D_2$) : $0.666 \times 300 = 200$mm
- Blade attack angle ($\alpha$) : 16 degree (assumed)
- Blade inlet angle ($\beta_1$) : 30degree
- Blade inner radius ($r_b$) : $r_b/D_1 = 0.157$ [8] $r_b = 47$mm
- Radius of center of runner to where blades are drawn ($R$) : $R/D_1 = 0.378$ [8] $R = 113$mm
- Number of blades : 24 (assumed)

Blade profile is depicted in Fig. 2.

![Blade profile creation in Pro-e](image)

Fig. 2: Blade profile creation in Pro-e

3.2 Nozzle Design

Various parameters considered for nozzle are as given below.

Nozzle design parameters
- Nozzle width ($B$) whereas $W$ is runner width : $B/W = 1$
- Nozzle Opening ($L$) : $L/D_1 = 0.254$ [8]
ICHPSD-2015

\[ L = 0.254 \times 300 = 76 \text{mm} \]

\[ \frac{r_0}{D_1} = 0.567 \] [8]

Admission Arc = 90 degree
Nozzle length from centre of runner = 170 mm (assumed)
Gap between Runner and nozzle lower surface = 2 mm

Fig. 3 shows the nozzle profile created in Pro-e.

**Fig. 3: Nozzle profile made in Pro-e**

The 2-D drawing of nozzle runner assembly is made as shown in Fig.4. It depicts the full geometry of cross flow turbine with dimensions.

**Fig. 4: Full geometry of cross flow runner with nozzle**
4. METHODOLOGY FOR CFD BASED DESIGN

In order to define the outline boundary at outlet, nozzle casing is created in pro-e 3D CAD software and then imported to Acusolve CFD software. In this software, Acuconsole is the subprogram of Acusolve which is used for meshing purposes and for defining all boundary conditions for surfaces, turbulence equation, type of analysis, mesh global attributes and material properties. The flow domains consist of nozzle walls, runner blades and solid walls (lower portion of turbine). First of all 3D model is created in pro-e 2.0 parametric, the acuconsole model was imported and positioned at the required central position.

The numerical analysis involves the runner, which has a rotating frame of reference while the nozzle and turbine casing has no rotating reference frame. These are considered as stationary walls.

Modeling of the flow problem is done in Acuconsole before meshing which includes creating reference frame for rotating flow zones or domains, assigning boundary conditions, initial conditions applying governing equations with appropriate turbulence model and choosing the flow materials (water) and their phase properties. The solution control and declaration of variables are done in this part.

The flow domain is meshing using the Acuconsole sub-program of the Acusolve software. The mesh priorities are given in following Table 1.

4.1 Grid Generation

One of the most cumbersome and time consuming part of the CFD is the mesh generation. Although for very simple flows, mesh generation is easy, it becomes very complex when the problem has many cavities and passages. Mesh generation is basically the discretization of the computational domain.

In the process of grid generation Acuconsole tool sub program of Acusolve was used. Global mesh attribute relative mesh size was used for all parts of turbine that help to reduce the converging time. Further in the area of higher gradients of analyzed parameters, higher density of grid can use to obtain the acceptable level of solution or fine meshing but it also increases the solver time. A graphical view of grid generation is shown in Fig.5. Mesh statistics obtained are as follows.

| Number of Nodes | = 79438 |
| Number of Elements | = 321560 |
| Number of Surfaces | = 92340 |
4.2 Mesh Properties

In numerical solutions, it is important to make sure that the mesh is of good quality so as to produce results that make sense with respect to the nature of physical process. Also, a good quality mesh reduces computational costs. Table 1 lists mesh property used in this study. Other mesh conditions are used as default value.

<table>
<thead>
<tr>
<th>Mesh size type</th>
<th>Relative</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relative mesh size</td>
<td>0.002m</td>
</tr>
<tr>
<td>Curvature angle</td>
<td>25 degree</td>
</tr>
</tbody>
</table>

4.3 Mesh Sensitivity

After obtaining a good quality of mesh, it is important to perform a mesh sensitivity analysis to make sure that the solution is mesh independent, as it has already stated. The sensitivity analysis involves the physical output variable to be monitored and recording its value with respect to change in number of mesh elements. The cut plan view of turbine is used to visualize the mesh sensitivity. Cut plane is helped to check the mesh structure inside the model. Fig. 6 shows the cut plan view of the cross flow turbine.
4.4 Boundary Conditions

The boundary conditions mean to define the problem or to specify the information of the flow variables that results in a unique solution. Defining boundary conditions involve identifying the locations of the boundaries (for example: inlet, outlet, walls, symmetry) as well as supplying information at the boundary. Poorly defined boundary conditions can have significant impact on solution.

In Acuconsole (pre-processor for Acusolve) the flow domain is defined. There is no solid domain involved in the study. The flow in this study is steady state hence Spalart Allmars turbulence model is chosen. The boundary conditions are specified in Acuconsole pre-processor and then the file is exported to solver.

In this case of computation, mass flow rate is assigned at the inlet boundary condition. A reference frame is created for rotating parts and default walls type with reference frame boundary conditions are defined for runner blades. Walls type with other default values boundary conditions are assigned at the nozzle walls and other all solid walls. The details of boundary conditions are given in Table 2.

Table 2: Details of Boundary Conditions

<table>
<thead>
<tr>
<th>Location</th>
<th>Assign boundary condition type</th>
<th>Boundary condition details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Inflow</td>
<td>Mass flow rate</td>
</tr>
<tr>
<td>Outlet</td>
<td>Outflow</td>
<td>Atmospheric pressure</td>
</tr>
<tr>
<td>Blades</td>
<td>Wall type</td>
<td>Wall boundary condition default details Reference frame</td>
</tr>
<tr>
<td>Nozzle walls and casing solid walls</td>
<td>Wall type</td>
<td>Wall boundary with default details</td>
</tr>
</tbody>
</table>
4.5 Turbulence Model

4.5.1 Spalart allmaras

The Spalart Allmaras model adds a single additional variable for a Spalart Allmaras viscosity and does not use any wall function; it solves the entire flow field. It is less memory intensive than the other models that solve the flow field in buffer layer. Its advantage is that it is quite stable and shows good convergence. It also takes less time in converging than other turbulence model.

4.6 Numerical Solution Approach

Solution phase is completely automatic. The FEA software generates the element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used by the subsequent phase (post-processing) to review and analyze the results.

The CFD flow solver used is ACUSOLVE, which is a General-purpose 3-dimensional, unstructured flow solver which uses the incompressible Reynolds-averaged Navier-Stokes equations. The solver is based on the finite element method to build spatial discretization of the transport equations.

Solver control and some parameter used in Acusolve numerical analysis was as discussed below.

4.6.1 Basic settings

- Convergence control set to 150 as maximum iteration
- Convergence area set to 0.001

Running Average output
- Time step frequency 1
- Field order set to 2

Restart output & Time Average output
- All settings are kept default

In the CFD model the mass flow rate is defined at the inlet and one reference frame is created with 300 RPM and this reference frame is assigned for the rotating blades. Based on the boundary conditions applied in the input parameters, the mass and momentum conservation equations were solved iteratively and various output parameters were generated. Torque (T) acting on the turbine is calculated based on the total moment acting on the rotating runner. In the Acu probe subprogram of Acusolve plotted the results was obtained in the graphical form. In the program various results such as moment, pressure, mass flux, velocity etc. based on giving the input parameters can be calculated. In the surface output panel blade z-moment is plotted. Based on the calculation, the value of torque is obtained about 150 N-m.
5. CONCLUSIONS

Cross flow turbine is considered one of the simplest turbine for micro hydro especially for low capacity hydropower plants. However, it has inherently low efficiency, which can be improved up to certain extent by modification in turbine. Under the present paper an attempt has been made to discuss CFD based design of 5.0 kW capacity cross flow turbine. The work carried out may be useful to analyze the performance of a cross flow turbine modified designs be proposed in future.

6. REFERENCES