Abstract

The low efficiency is the basic inspiration to improve the existing design of cross flow turbine. It is observed that the efficiency of cross flow turbine can be increased by introducing a proper guide mechanism to guide the flow inside the turbine runner. The objective of the present investigation is to develop a CFD simulation strategy for cross flow turbine for modification in the existing runner. Critical design points in cross flow turbine are the blade profile, flow guide mechanism and number of blades which play very important role in predicting the torque. The torque generated in the turbine is a direct measure of turbine performance. Other design parameters include flow velocity and efficiency. The simulation was performed using Altair’s CFD product suite AcuSolve.

1. Introduction

The capacity of micro hydro power plant up to 100 kW is considered under micro hydro power 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. Cross-Flow turbine has been considered technoeconomically viable for such sites. Cross flow type runner can be fabricated locally, which results in the poor efficiency. The vanes can be made even from the pipes cut along the length. Also this kind of runner is suitable for low discharge and high head conditions, which is a common case in the hills. The problem with the fabrication of cross flow turbine runner at the local level is that its efficiency decreases due to lack of proper design and fabrication. The low efficiency is the basic inspiration to improve the existing design of cross flow runner. It is observed that the efficiency of cross flow turbine can be increased by modification in turbine runner. Various researchers attempted to improve the efficiency of cross flow turbine. Khosrowpanah S, [1] carried out a study of cross flow turbine experimentally by varying the number of blades, the runner diameter, and the nozzle entry arc under flow-head variations. Desai and V.
Rangappayya [2] carried a parametric study of cross flow turbine analysis to find out the key parameters influencing the turbine efficiency and a theoretical analysis of turbine performance. Parametric study on the performance of cross flow turbine was done by C.B. Joshi et al. [3]. Effect of draft tube size on the performance of a cross-flow turbine was studied by H. Reddy et al. [4]. Study of the nozzle flow in a cross flow turbine studied by N.H. Costa Pereira and J.E. Borges [5]. The development of the high speed computer and the evolution of the computational fluid dynamics (CFD) have a great influence on the engineering design and analysis of the turbo machinery. In the past decades, permitted by the increase in the capability of coming with complex geometry and complex flows and reduction in computational time and costs, the CFD methodology has emerged to become an efficient approach for collecting information to improve engineering design of turbo machinery [6]. HyperWorks is a superior product design and virtual testing software. It provides high-speed and high-quality Meshing. The modeling process of complex geometries is simplified with automatic and semi-automatic shell, tetra, and hexa meshing capabilities of HyperMesh. It has a powerful solver, AcuSolve, which is very robust, speedy and accurate. In this paper, the fundamentals of AcuSolve and its application are presented in the cross flow turbine.

2. Process Methodology

The methodology is adopted in four steps. The first step is the preparation of geometry as per the standard design, second step is meshing the geometry, third step provides the boundary conditions as per data available at the site and last step is the simulating the model and observe the results in the post processor. The numerical investigation has been performed in HyperWorks 12.0 software with its subprograms are namely: AcuConsole (Pre processor), AcuSolve (solver), AcuProbe and AcuFieldView (post processor).

2.1 Modeling and Grid Generation

The runner and nozzle are the main parts of a cross flow turbine. The complete model of cross flow turbine has been developed in the modeling software Pro-E. A cross flow turbine having a capacity of 5.0 kW has been considered, the details of which are given in Table I.

Further, the model was imported in to HyperMesh for meshing. Mesh generation basically refers to the discretization of the computational domain. It is one of the most cumbersome and time consuming
part of the CFD analysis. Although for very simple flows, mesh generation is easy, it becomes very complex when the problem has many cavities and passages.

<table>
<thead>
<tr>
<th>Table I. 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>Turbine speed (N)</td>
</tr>
</tbody>
</table>

HyperMesh tool, a sub program of HyperWorks, has been used for grid generation. Global mesh attribute relative mesh size has been used for all parts of turbine that help to reduce the converging time. In the area of higher gradients of analyzed parameters, higher grid density or finer meshing can be used to obtain acceptable levels of solution accuracy, but it also increases the computation time. A graphical view of grid generation is shown in Fig. 1. The statistics of the grid generated are given in Table II.

![Fig. 1 Mesh generation](image)

<table>
<thead>
<tr>
<th>Table II Mesh Statistics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Nodes</td>
</tr>
<tr>
<td>Number of Elements</td>
</tr>
<tr>
<td>Number of surfaces</td>
</tr>
</tbody>
</table>
2.1.1 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 is already stated. The sensitivity analysis involves monitoring the physical output variable and recording its value with respect to change in number of mesh elements. The cut plane view of turbine is used to visualize the mesh sensitivity. Cut plane view is useful for checking the mesh structure inside the model. The cut plane of geometry has been shown in Fig. 2.

![Fig. 2 Cut Plane view of Model](image)

2.2 Solver Setup

The model after being meshed has been solved using the AcuSolve tool of HyperWorks. The meshing file is exported from HyperMesh to AcuConsole, where the boundary conditions are defined and the solving parameters are selected. The quality and sensitivity of the mesh has been checked before proceeding with the solution.

2.2.1 Boundary Conditions

Defining boundary conditions involve identifying the locations of the boundaries (such as inlet, outlet, walls, etc.) as well as supplying the information at the boundary. Poorly defined boundary conditions can have significant impact on the solution. The flow domain has been defined in AcuConsole.
The boundary conditions have been specified in AcuConsole preprocessor and then the file has been exported to the solver.

Mass flow rate has been defined at the inlet and pressure at the outlet. A reference frame has been created with 300 rpm and has been assigned to the rotating blades. A summary of the boundary conditions is given in Table III.

<table>
<thead>
<tr>
<th>Location</th>
<th>Boundary Condition Type</th>
<th>Boundary Condition Detail</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>

2.2.3 Turbulence Model

Steady flow has been considered in this study, hence Spalart Allmars turbulence model has been chosen. The SA model adds a single additional variable for eddy viscosity and uses a wall function to map the near wall regions based on the Y+ Value. Its advantage is that it is quite stable and shows good convergence. It also takes less time in converging than other turbulence model.

2.2.4 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, namely AcuSolve, which is a general purpose 3-dimensional, unstructured flow solver, uses the incompressible/ weekly compressible Reynolds-averaged Navier-Stokes equations. The solver uses finite element method to build spatial discretization of the transport equations. Based on the boundary conditions, the governing equations are solved iteratively up to the tolerance limit and the various output parameters are generated.

3. Results And Discussions

The results obtained can be viewed in the graphical form in the AcuProbe subprogram of AcuSolve. In the program the value of the various output parameters such as moment, pressure, mass flux, velocity, etc. can also be obtained. The torque acting on the blades can be calculated based on the total moment.
The vector plot of the pressure function of cross flow turbine is shown in Fig 3. It provides the information about direction of the fluid flow. It can be clearly seen that vectors enter at the periphery of the runner and then cross the shaft before coming out of the runner. Fig. 4 shows the variation of pressure across the turbine. The pressure drop in the turbine from the nozzle to the outflow channel depicts the gain in momentum.
Fig. 5 Pressure variation over the runner blades

Fig. 6 Z-moment at the runner
Further, the variation in pressure at the runner blades is shown in Fig. 5. It can be observed that maximum pressure drop occurs corresponding to the first stage of the blade as compared to second stage. It clearly shows the maximum energy has been transferred at the first stage of the turbine. Fig. 6 illustrates graph of the moment at z axis as a function of time for the runner. Based on the result, the value of z-moment is obtained about 135 Nm. It indicates the torque produced by the runner. The efficiency has been predicted of the turbine about 60.0 %.

4. Conclusion

In this paper, a CFD analysis of fluid flow in a cross flow turbine has been carried out. Based on the CFD analysis, it has been found that the computed torque is well predicted. AcuSolve simulation results have prominently contributed for understanding the imprecisely the fluid flow in the cross flow turbine. The solver is highly stable and is capable of efficiently solving unstructured finite element meshes with high aspect ratio and badly distorted elements.

ACKNOWLEDGEMENTS

We acknowledge the support rendered by Mr. Rishikesh Sanjiv Hungund, Asst Manager Tech Support DesignTech Systems Ltd, New Delhi towards submission of a research paper at the Altair Technology Conference who was commendable and instrumental in raising the caliber of the work done to a level where it was worthy enough to result in a paper publication.

REFERENCES
