DESIGN AUTOMATION AND CFD ANALYSIS OF DRAFT TUBE FOR HYDRO POWER PLANT

1GUNJAN B. BHATT, 2DHAVAL B. SHAH, 3KAUSHIK M. PATEL

1Post Graduate Student, Mechanical Engg. Dept., Institute of Technology, Nirma University, India
2Assistant Professor, Mechanical Engg. Dept., Institute of Technology, Nirma University, India
3Professor, Mechanical Engg. Dept., Institute of Technology, Nirma University, India
Email: 1gunjanbhatt23@gmail.com, 2dbshah@nirmauni.ac.in, kaushik.patel@nirmauni.ac.in

Abstract—The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. The shape and velocity distribution at the inlet are, in next turn, two main factors that affects the performance of the draft tube. Traditionally, the design of this component has been based on simplified analytic methods, experimental rules of thumb and model tests. In this paper, an attempt has been made for design automation of modeling of draft tube using Excel spreadsheet and Creo parametric software. In the last decade or two, the usage of computational fluid dynamics (CFD) has dramatically increased in the design process and will continue to grow due to its flexibility and cost-effectiveness. A CFD-based design search can further be aided with a robust and user-friendly optimization framework work theory and engineering. In this paper, the CFD analysis of draft tube has been performed and results for the same are compared with experimental reading and which are found within the limit.

Index Terms—CFD analysis, draft tube, pressure, stream line.

I. INTRODUCTION

The utilization of the hydraulic force in the domain of the electricity production has long been majority; it has started in antiquity with mills of water. The techniques permitting the exploitation of hydroelectric resources have benefited important progress during the twentieth century, in the scope of projects construction of the hydroelectric center of great speed. Through their size, their precision and their efficiency, the equipment of these hydroelectric center and especially hydraulics turbines arrived in first plan of realization. The hydraulic turbine is a mechanic dispositive which is used to transform potential energy and the kinetic energy of water, in mechanic energy. This will then be transformed into electric energy by an alternator. There exist two categories of hydraulics turbine. The turbine of action, which do not constitute a draft tube and function with the kinetic energy of water, and the turbine of reaction, which function with the pressure difference and the kinetic energy pressure.

With the increasing cost of energy and the high demand of green energy, hydraulic turbine of thin height of falls such as Francis and Kaplan turbines, are those targeted as being economically profitable. They are constituted of distributor, volute, of runner and draft tube. The draft tube permits the recuperation of excess water kinetic energy coming from the runner and converts it into energy of static pressure.

Many studies on the draft tube flow have been done. Marjavaara [1], carried out a numerical study to show that the draft tube have an important rule on the global efficiency of a hydraulic turbine. According to Cervantes et al. [2], draft tubes are of great interest for turbines of thin height of fall like Kaplan turbines, since the draft tube efficiency increases with the decreasing of the height of fall. Anderson [3], had done an experimental study in the draft tube cone and demonstrated that for small height of fall and high output, losses in draft tube are considerably high and can go up to 50%. Labrecque [4], did a study on the conception of the axial turbine and demonstrated that the augmentation of the performance of hydraulic turbine pass by a good knowledge of the flow in the turbine.

II. DESIGN AUTOMATION OF DRAFT TUBE

As per the client’s specification, dimensions for draft tube have been decided and 2D drawing for the same has been prepared as shown in Fig. 1.
Here Microsoft Excel and Creo are integrated by Excel Analysis tool which acts as an interfacing media. Excel analysis tool transfers the spreadsheet data to the Creo database. An excel spreadsheet which contains feature names and respective dimensions of the draft tube has been prepared as shown in Fig 2.

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FEATURE NAME</td>
<td>DIMENSION</td>
</tr>
<tr>
<td>2</td>
<td>CENTRE DIST 1</td>
<td>1000</td>
</tr>
<tr>
<td>3</td>
<td>CENTRE DIST 2</td>
<td>850</td>
</tr>
<tr>
<td>4</td>
<td>CENTRE DIST 3</td>
<td>620</td>
</tr>
<tr>
<td>5</td>
<td>FILLET 1</td>
<td>500</td>
</tr>
</tbody>
</table>

Fig 2: Excel Spreadsheet

An Excel Analysis from Analysis Tab have been selected in Creo as shown in Fig 3 to transfer data from Excel Spreadsheet to Creo.

Fig 3: Excel Analysis in Creo

In step 1, the model of draft tube has been converted into STEP file and this step file has been imported in ANSYS. After importing model in ANSYS cavity model of draft tube has been created using fill command shown in fig 5.

Fig 5: cavity model of draft tube

In step 2, the meshing of this draft tube model is done. In meshing CFD mesh type is selected and fine meshing is done by using ten node tetrahedral elements shown in fig 7. The reason for selecting this element is that it gives the good meshing on curvature parts. Meshing model of draft tube is shown in fig 6. In meshed model 12,330 nodes and 66,821 elements were created.

Fig 6: Meshing of draft tube

III. CFD ANALYSIS

Here ANSYS workbench is used for CFD analysis of draft tube. For CFD analysis following steps are performed. In step 1, the model of draft tube has been converted into STEP file and this step file has been imported in ANSYS. After importing model in ANSYS cavity model of draft tube has been created using fill command shown in fig 5.

Fig 7: Ten node tetrahedral element

User can update the model just by modifying the sheet. This takes comparatively very less time to generate complex part models with respect to generating them individually. This technique is more suitable and simpler than any other technique like VB interfacing.
The boundary condition for draft tube analysis have been applied. In boundary condition inlet mass flow rate is given 1000 kg/sec and outlet boundary condition is given 1atm as shown in fig 8 and fig 9.

Now next step is run the CFX solver to get the result. After that last step is analyses and visualize the result in post processor. Here in post processor, the pressure and velocity distribution have been visualized and same are as shown in fig 10 and fig 11.

Fig 12 and fig 13 shows the pressure at inlet(-2.100×10^5 Pa) and pressure at outlet(1.071×10^5 Pa)respectively. The streamline velocity contour has been shown in fig. 14.
IV. COMPARISON WITH EXPERIMENTAL READING

The pressure and velocity at inlet and outlet of draft tube has been measured by experimental procedure. The same results have been compared with ANSYS analysis results and % difference has been found as given in table 1, which shows both results are in good agreement with each other.

Table 1: comparison between ANSYS and practical reading

<table>
<thead>
<tr>
<th></th>
<th>INLET PRESSURE</th>
<th>OUTLET PRESSURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSYS RESULT</td>
<td>-2.100 x 10⁵ Pa</td>
<td>1.071 x 10⁵ Pa</td>
</tr>
<tr>
<td>PRACTICAL READING</td>
<td>-1.99 x 10⁵Pa</td>
<td>1.12 x 10⁵Pa</td>
</tr>
<tr>
<td>% DIFFERENCE</td>
<td>5.23 %</td>
<td>4.38 %</td>
</tr>
</tbody>
</table>

CONCLUSION

ANSYS is very important software for analysis purpose and creo is very powerful software for making parametric modeling. So one can easily change 3D model dimension in less time for optimization purpose. The application of ANSYS is wide in engineering field. From above analysis it has been found that ANSYS result give the good agreement with experimental reading. By design automation one can reduce the time and cost of modeling and drafting. By analysis results one can predict almost nearer results of pressure and velocity profile without carrying higher cost experimental work.

REFERENCES