CFD approach for design optimization and validation for axial flow hydraulic turbine

Vishnu Prasad, V K Gahlot* & P Krishnamachar
Department of Civil Engineering, Maulana Azad National Institute of Technology, Bhopal 462 051, India

Received 30 July 2008; accepted 3 June 2009

The conventional method to assess turbine performance is its model testing which becomes costly and time consuming for several design alternatives in design optimization. Computational fluid dynamics (CFD) has become a cost effective tool for predicting detailed flow information in turbine space to enable the selection of best design. In the present paper, 3D real flow analysis in an experimentally tested axial flow turbine has been carried out and different flow parameters are computed at three operating regimes to find the best operating regime. The computed efficiencies are critically compared with experimental values and found to bear close comparison.

Keywords : Computational fluid dynamics, Efficiency, Lift, Blade circulation, Degree of reaction

In axial flow turbine, water passes through the series of blade rows and changes its direction from radial to axial. Runner is the most important component of the turbine and its blade profile is designed at different sections from hub to casing to get the best performance. The rotation of runner and operation of turbine either below or above the rated conditions\textsuperscript{1,2} cause variation of flow parameters from hub to tip. Hence, actual flow pattern in turbine space deviates from the simplifying assumptions made in design thus affecting the turbine performance. The experimental testing of turbine models at different operating regimes on specially designed test rigs is the conventional approach to assess the performance. But this approach provides global performance at varying operating conditions and does not give detailed information about the flow behaviour and variation of local design parameters like blade circulation, lift and degree of reaction. The model fabrication and testing for any change made in design make this method costly and time consuming.

The combination of advanced numerical techniques and computational power has lead to computational fluid dynamics (CFD). It is an efficient and inexpensive tool to make internal flow predictions to good accuracy and, any sort of flow problems can be detected and further improvements can be made on the geometry of turbine components. It has made possible to obtain a significant enhancement in efficiency\textsuperscript{3,6} of the hydro turbine. CFD can be used to check efficacy of alternate designs\textsuperscript{7,8} of turbines for optimization before final experimental testing of selected designs is resorted. However, in order to prove reliability of these tools for application to turbines, validation\textsuperscript{9,11} with known experimental results is required. In present work, 3D viscous flow simulation with SST $\kappa-\omega$ turbulence model is carried out in an experimentally tested model of an axial flow hydraulic turbine using ANSYS CFX10 software. The variations of flow parameters from hub to tip of runner are presented in graphical form and average value of cascade parameters are computed at different operating regimes. The computed efficiencies have been compared with experimental values at three regimes for validation of simulation results.

Geometric Modeling and Boundary Conditions

The axial flow turbine consists of casing, stay ring, distributor, runner and draft tube. The energy transfer takes place in runner hence, present work is focused on runner only and therefore, analysis is carried from stay ring inlet to draft tube outlet where proper boundary conditions can be applied. There are 12 stay vanes, 28 guide vanes and 4 runner blades in the model being analyzed. The blade rows of stay ring,
distributor and runner are axi-symmetric and therefore, only single runner blade assembly is modeled for simulation using periodicity to minimize the total size of mesh. The draft tube is fully modeled because of no symmetry about any axis. Each component is modeled separately and then assembled through proper interfaces. The guide vane is modeled for three different openings. The flow parameters at inlet and exit of runner blade with velocity triangles are defined in Fig. 1. The meridional view with dimensional parameters and assembled 3D view are shown in Figs 2 and 3. The unstructured tetrahedral mesh is generated in Ansys ICEMCFD for all the domains. The $y^+$ varies between 24 to 186, which is the acceptable range for automatic wall function treatment in boundary layer in SST $\kappa$-$\omega$ turbulence model. The maximum values other mesh quality parameters like face angle (163.56), edge length ratio (28.758), connectivity number (48) are within the acceptable limits of Ansys CFX10.

The meshing for stay vane, guide vane, runner and draft tube are given in Figs 4-7. The summary of mesh data of each component is given in Table 1. The magnitude of mass flow and its direction are specified at stay vane inlet as inlet boundary condition and reference pressure is specified at outlet of draft tube as outlet boundary condition. The rotational speed of runner is specified and other two blade rows are set stationary. All boundary walls are assumed smooth with no slip.

**Experimental Model Testing**

The experimental testing of axial flow turbine model at reduced scale had been carried out on specially designed test rig as per IEC codes. The geometric specifications of tested axial flow turbine model are given below:

<table>
<thead>
<tr>
<th>Component</th>
<th>No. of nodes</th>
<th>No. of elements</th>
<th>Type of element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stay vane</td>
<td>24811</td>
<td>126527</td>
<td>Tetrahedral</td>
</tr>
<tr>
<td>Guide vane</td>
<td>16212</td>
<td>78051</td>
<td>Tetrahedral</td>
</tr>
<tr>
<td>Runner</td>
<td>35267</td>
<td>180868</td>
<td>Tetrahedral</td>
</tr>
<tr>
<td>Draft tube</td>
<td>279151</td>
<td>1540907</td>
<td>Tetrahedral</td>
</tr>
</tbody>
</table>

**Fig. 1** – Velocity triangles at inlet and outlet of runner blade

**Fig. 2** – 2D meridional view of annulus space of turbine

**Fig. 3** – Assembled 3D turbine model
In the model testing, variations of global parameters like discharge, speed and power have been observed. However, it is very difficult, rather impossible to measure the local parameters like velocities and flow angles at runner because of its rotation. The efficiencies of axial flow turbine model at three regimes, i.e., guide vane openings of 50°, 40° and 35° from experimental results are 90.86%, 92.06% and 91.59% respectively.

**Computation of Flow Parameters**

The numerical analysis gives pressure and velocity distributions and the non-dimensional parameters are computed for presentation of results. The velocity components are divided by spouting velocity ($\sqrt{2gH}$) to get specific (non-dimensional) values of corresponding velocity. The following formulae are used for computation of different parameters:

1. **Pressure coefficient**
   \[ C_p = \frac{P - P_2}{\frac{1}{2} \rho W_2^2} \]  
   … (1)

2. **Velocity coefficient**
   \[ C_s = \frac{W}{W_2} \]  
   … (2)

3. **Flow deflection**
   \[ \varepsilon = \beta_1 - \beta_2 \]  
   … (3)

4. **Degree of reaction**
   \[ \phi = \frac{W_z^2 - W_1^2}{2gH} \]  
   … (4)

5. **Circulation coefficient**
   \[ \tau = \frac{t(C_{\psi 1} - C_{\psi 2})}{D\sqrt{2gH}} \]  
   … (5)

6. **Lift coefficient**
   \[ C_L = \frac{2}{I} \sin \beta_n (\cot \beta_2 - \cot \beta_1) \]  
   … (6)

7. **Runner energy coefficient**
   \[ \phi = \frac{gH_x D^3}{Q^2} \]  
   … (7)

8. **Total energy coefficient**
   \[ \psi = \frac{gHD_y^4}{Q^2} \]  
   … (8)
Total head

\[ H = \frac{TP_{SV} - TP_{DTE}}{\gamma} \]  \quad \ldots (9)

Head utilized by runner

\[ H_R = \frac{(TP_1 - TP_2)}{\gamma} - H_{FR} \]  \quad \ldots (10)

Hydraulic efficiency

\[ \eta_H = \frac{H_R}{H} \times 100 \]  \quad \ldots (11)

Draft tube efficiency

\[ \eta_{DT} = \frac{2gH_{DTE}}{C_2^2} \times 100 \]  \quad \ldots (12)

Results and Discussion

The numerical simulation of axial flow hydraulic turbine using Ansys CFX10.0 is validated with experimental results at three guide vane openings near the points of maximum efficiency. The comparison of computed and experimental values of efficiency is given in Table 2. The values of SF and QF are kept same in both experimental and numerical simulation at individual operating regime. It is seen from Table 2 that peak efficiency regime obtained in both numerical simulation and experiment is same. The computed efficiencies at different regimes are in close agreement with the experimental values. The differences in efficiencies at off-peak regimes obtained between simulation and experimental values may be ascribed to (i) errors in discretization of governing equations and flow domain, (ii) the losses not accounted for fully and precisely and (iii) not considering complete assembly of casing, stay vanes, guide vanes and runner in CFD analysis. There can be instrumentation and human errors in the experimental investigation.

Further, numerical analysis has been carried out for three different regimes of operation as given in Table 3. The three regimes are selected such that one regime is near the best operating point (\( \alpha = 40^\circ \)) to enable the comparison of results at design and off-design conditions. The mass averaged values of specific velocities and flow angles given in Table 4 indicate that all velocity triangle components are affected by guide vane setting except the flow angles at runner outlet.

The specific meridional velocities (\( c_m \)) at runner inlet and outlet are nearly same for each guide vane and it is indicative of axial flow. The specific whirl (\( c_u \)) and absolute (\( c \)) velocities decrease from inlet to outlet due to energy extraction by the runner. The specific relative velocity (\( w \)) increases from inlet to outlet of runner because of reduction in pressure in the form of reaction contributing to energy extraction by runner. The values of cascade parameters in Table 5 indicate that flow deflection, degree of reaction, lift and drag and energy losses are minimum at \( \alpha = 40^\circ \) while blade circulation decreases with decrease in guide vane (GV) opening. The efficiency is more at \( \alpha = 40^\circ \) and decreases on either side of this GV opening and thus indicative of best efficiency operating regime. The computed losses mentioned in Table 6 show that total loss is minimum at \( \alpha = 40^\circ \) and hence maximum efficiency but loss in not
minimum in all components. It is observed that main components affecting efficiency of turbine are runner and draft tube.

The pressure and velocity from leading edge (LE) to trailing edge (TE) at mid span of runner blade are shown in Figs 8 and 9 respectively. The blade loading at any point increases with increase in guide vane GV opening and there is smooth variation except at hub region due to hub curvature.

<table>
<thead>
<tr>
<th>Component</th>
<th>(\alpha = 50^\circ)</th>
<th>(\alpha = 40^\circ)</th>
<th>(\alpha = 35^\circ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stay ring loss (%)</td>
<td>0.638</td>
<td>0.740</td>
<td>0.642</td>
</tr>
<tr>
<td>Distributor loss (%)</td>
<td>0.766</td>
<td>1.302</td>
<td>2.037</td>
</tr>
<tr>
<td>Runner loss (%)</td>
<td>7.112</td>
<td>4.151</td>
<td>3.782</td>
</tr>
<tr>
<td>Draft tube loss (%)</td>
<td>7.845</td>
<td>2.532</td>
<td>3.362</td>
</tr>
<tr>
<td>Total loss (%)</td>
<td>16.361</td>
<td>8.725</td>
<td>10.496</td>
</tr>
<tr>
<td>Draft tube efficiency (%)</td>
<td>52.626</td>
<td>78.246</td>
<td>73.87</td>
</tr>
</tbody>
</table>

The increase in velocity with increase with GV opening is because of discharge increases with GV openings. The blade circulation in Fig. 10 depicts that it has maximum value near to quarter of span and decreases towards hub and tip indicating decrease in difference of whirl velocities at tip side and circulation is less at hub due to small pitch. It decreases with decrease in GV opening at any span. The flow deflection decreases from hub to tip as seen in Fig. 11. This indicates the decrease in inlet flow angle towards tip as outflow angles are constant for all guide vanes.

The degree of reaction decreases with decrease in GV at faster rate near hub region as compared to tip region as shown in Fig. 12. This indicates more change in pressure in hub region at larger GV opening. The lift variation in Fig. 13 depicts that it...
decreases from hub to tip and its variation becomes steeper with decrease in GV opening. This variation is attributed to the variation of flow deflection and pitch-chord ratio.

The streamline plots on the runner blade at mid span for three regimes are shown in Fig. 14. It is seen that flow is not smooth at $\alpha = 50^\circ$ and with very little shock in $\alpha = 40^\circ$ and very smooth in case of $\alpha = 35^\circ$.
and hence accordingly loss in runner is maximum at $\alpha = 50^\circ$ and minimum at $\alpha = 40^\circ$ as shown in Table 6.

The effect of boundary layer at the blade surface can be clearly seen at $\alpha = 40^\circ$. Similarly the pressure contour variation on suction and pressure side is more smooth in case of $\alpha = 40^\circ$ as compared to other operating regimes as shown in Fig. 15.

The 3D plots for streamline and pressure contours at $\alpha = 40^\circ$ are shown in Figs 16-18. The streamline and pressure contours for 3D flow in stay vane and guide vane domains shown in Fig.16 depicts that velocity increases and pressure decreases as flow moves from stay vane to guide vane domain. The velocity is more and pressure is less on suction side (lower surface) and velocity is less and pressure is more on pressure side (upper surface) of runner blade as seen in Fig. 17. The streamline and pressure contour pattern within draft tube shown in Fig. 18 indicate that low velocity zone is formed at concave side of bend. The velocity decreases and pressure
increases from inlet to exit of draft tube due to increase in flow area.

Conclusions
The validation of numerical simulation with experimental results shows similar pattern for efficiency variation. The maximum efficiency regime indicated by both the approaches is nearly same. The slight difference in computed and experimental values of efficiency can be because of some errors in numerical simulation which are mainly due to discretization of domain and differential equations and also instrumentation and human error during experimental investigations. The total computed loss is minimum at the point of maximum efficiency. The streamline and pressure contour plots in different components confirms with the actual flow behaviour in axial flow turbine. The best operating regime can be easily identified from computed flow parameters, losses and flow pattern from numerical simulation. Hence, it is concluded that CFD approach can be used to study the flow pattern inside the turbine space and to optimize the design by different combinations of the design parameters and geometry at low cost in lesser time. Finally, the performance of optimized design need to be verified through model testing. This procedure will minimize time and the amount spent in development and optimization of hydraulic turbines.

Nomenclature

\[ L = \text{blade chord (m)} \]
\[ P = \text{static pressure at any point on blade surface profile (Pa)} \]
\[ Q = \text{discharge through turbine (m}^3/\text{s}) \]
\[ TP = \text{total pressure at runner (N/m}^2\text{)} \]
\[ TP_{SVI} = \text{total pressure at stay vane inlet (N/m}^2\text{)} \]
\[ TP_{DTE} = \text{total pressure at draft tube outlet (N/m}^2\text{)} \]
\[ T = \text{pitch of runner blades (m)} \]
\[ W = \text{relative velocity at any point of blade surface (m/s)} \]
\[ \alpha = \text{guide vane angle from tangential direction (°)} \]
\[ \beta = \text{relative flow angle (°)} \]
\[ \beta_m = \text{mean relative flow angle(°)} \]
\[ \gamma = \text{specific weight of water (N/m}^3\text{)} \]
\[ \rho = \text{mass density of water (kg/m}^3\text{)} \]

subscripts 1 and 2 denote values of parameters at inlet and outlet of runner respectively

References
1. Raabe I J, Hydro Power- The design, use and function of hydro mechanical hydraulic and electrical equipment (VDI-Verlag, GmbH, Dusseldorf ), 1985