ISSN 2229-5518

Numerical Simulation for Unsteady Flow Analysis of Kaplan Turbine

Vaibhav Chandrakar[1], Dr. Ruchi Khare[2]

[1]M. Tech (Hydro Power), MANIT
Bhopal 462051, India
vaibhavchandrakar47@gmail.com

[2]Asst. Professor, Dept. of Civil Engg, MANIT
Bhopal 462051, India
ruchif1@yahoo.com

Abstract: The presence of stationary and rotating blades in Kaplan turbine makes the flow complex in the turbine space. Actual flow inside the turbine passage is highly fluctuating. Model testing provides the variation of global hydrodynamic design parameters and it is impractical to test different design variants due to cost and time. The flow and pressure distribution inside the turbine can be analyzed more accurately through numerical simulation under unsteady flow condition. The courant number which is associated with time step variation directly signifies the unsteady flow conditions. The approximate solution of various non-linear partial differential equations governs the nature of flow. In the present work the overall performance of a Kaplan turbine is studied under unsteady flow conditions. The analysis is carried out at a fixed runner blade angle and three different guide vane openings by using ANSYS CFX software.

Keywords: unsteady flow, Kaplan, CFD, numerical simulation, courant number.

1. Introduction

Kaplan Turbine is an Axial Flow Reaction Turbine. The turbines are designed based on simplifying assumptions. But the actual flow inside the turbine space is not as per assumptions and hence turbine performance differs from design conditions. It is therefore, necessary to predict the actual turbine performance before making prototype. The conventional method to predict the turbine performance is testing of turbine model, which is a model of prototype at reduced scale and fulfils the hydraulic similitude conditions. The model construction and testing is costly and time consuming when several modifications in the design are needed. The model testing may provide only global performance characteristics and it is difficult to get characteristics for individual component.

With the growth of numerical methods and computational power now it is possible to carry out the numerical simulation in complex turbine space by solving non-linear partial differential equations governing the flow. The flow inside the turbine is complex and turbulent in nature. In this paper, numerical simulation of an experimentally tested axial flow hydraulic turbine model is carried out under unsteady flow conditions.

Different researchers have studied the effect of unsteady flow condition on the efficiency of turbine, Ruprecht et al. [2010][carried out the unsteady flow simulation in an entire Francis turbine from spiral case to the draft tube . They used unsteady flow simulation for a Francis turbine with 24 stay vanes and 13 runner blades and showed that the simulation are feasible, meaningful and quantitatively accurate results concerning dynamical forces and loading can be expected by this type of simulation. Therefore more accurate solutions on finer grids are in progress, applying a single time step for all the problems results in extensive requirements of computer. Khalifa et al. [2013] and Hyen-Jun Choi et al. [2013] also carried out the transient flow simulations for axial and mixed flow turbines to study its performance at different operating conditions. In the present paper an attempt has been made to simulate a low head axial flow turbine, and to study its performance in off design and rated conditions. The results obtained for efficiency and output are compared with experimental results at certain boundary conditions and found to have good agreement with experimental results.

2. Specifications of turbine

The numerical simulation has been carried out for an axial flow Kaplan turbine. The geometric dimensions of turbine model are given below:

Type of turbine	Vertical axial flow Kaplan turbine
Type of casing	Spiral type
Type of draft tube	Elbow type
Diameter of turbine runner	0.400 m
Pitch circle dia of guide vanes	0.480 m
Number of guide vanes	28
Number of stay vanes	12

3. GEOMETRIC MODELLING

The 3-D geometry of turbine space, profile of stay vane, guide vane and runner and draft tube is required for numerical flow simulation of turbine. The geometrical modeling of the complete turbine in a single domain is very complex. Therefore, different part of turbine i.e. casing, stay ring, guide vane runner and draft tube are modelled separately. The 3-D modeling is done in NX software and meshing is done by using ANSYS ICEM CFD. All domains are transferred to CFX and coupled with each other through interfacing. The unstructured tetrahedral meshing is done in each flow domain except at the runner domain. The mixed type meshing i.e. the layer of structured hexahedral meshing is applied at runner blade surface and tetrahedral meshing is done at remaining part of runner.

The reference pressure was taken as 1 atm., The numerical analysis was done by applying mass flow rare at inlet and pressure at outlet.

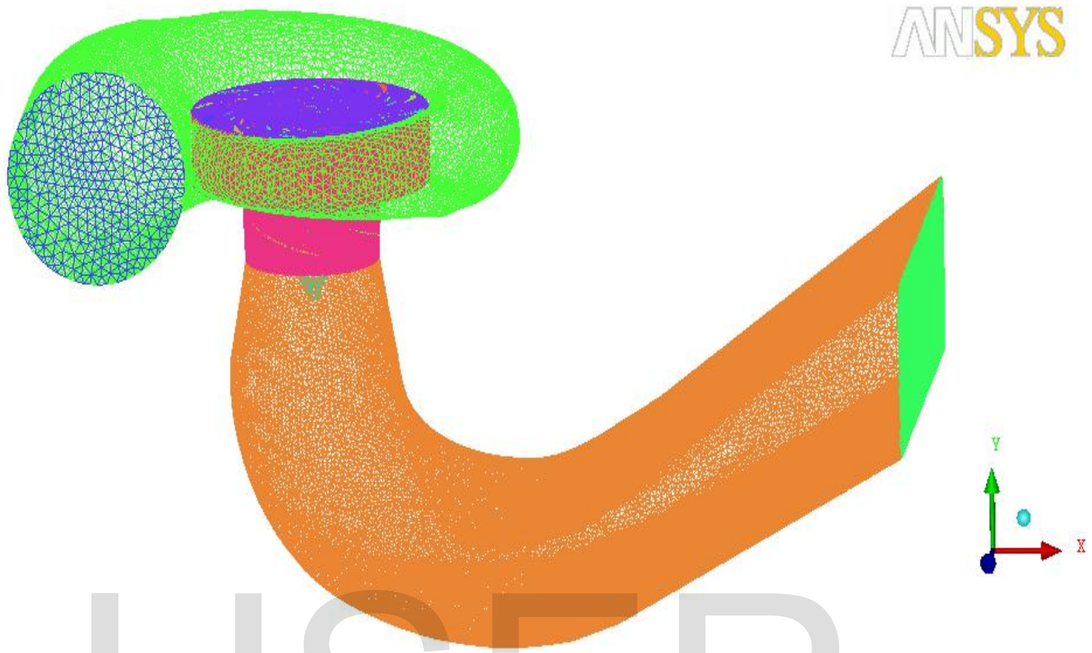

Fig. 1 - Meshing of complete assembly

4. Formulae used

Net head across the turbine $H = \frac{TP_1 - TP_2}{\rho\delta}$

Head utilized by the runner $H_R = \frac{(TP_1 - TP_2)}{\gamma} - H_{LR}$

Time step corresponding to 1° runner rotation

$$\Delta t = \frac{60}{360 \times N}$$

Input power $P_{in} = \rho g Q H$

Output power $P_{out} = \frac{2\pi NT}{60}$

Hydraulic efficiency $\eta = \frac{P_{out}}{P_{in}} * 100\%$

5. Results

The numerical flow simulation of Kaplan turbine under unsteady flow condition is done by using ANSYS CFX 14 software. The boundary conditions are applied as mass flow rate at the inlet of casing and static pressure at the outlet of draft tube . The performance of Kaplan turbine is analyzed at three guide vane opening i.e. 35º, 40º, 50º for four time steps corresponding to 0.25° 0.50° 1.0° and 1.5° runner rotations , at each guide vane opening. . The total simulation is carried for 90° of Kaplan runner rotation. The variation of computed flow parameters are presented in the graphical and tabular form.

5.1 Common input parameters

The numerical simulation for any flow domain requires 3-D geometry of flow space, boundary conditions, nature of flow and properties of fluid, some parameters are required to be specified for the numerical simulation depending upon potential or viscous flow analysis. The common parameters with their values used during analysis are given in Table 1

5.2 Validation of results

The efficiency obtained from numerical simulation for different guide vane openings under steady flow condition are compared with experimental results as shown in table 2. The efficiency obtained in case of numerical simulation is slightly higher than the experimental results because all losses may not be incorporated in numerical simulation.

Table 1: Details of boundary conditions

Boundary wall	smooth with no slip
Input boundary condition	mass flow rate specified as 0.525 m^3/s for 35° guide vane opening 0.620 m^3/s for 40° guide vane opening 0.714 m^3/s for 50° guide vane opening
Outlet boundary condition	specification of reference pressure at draft tube outlet as 0 atm.
Stationary blade rows	stay ring and guide vanes
Rotating blade row	runner with rotational speed specified as 1050 rpm for 35° guide vane opening 1150 rpm for 40° guide vane opening 1375 rpm for 50° guide vane opening
Type of interfaces	Fluid-Fluid
Interface model	General connection
Pitch change	Automatic, GGI Connection
Turbulence model	SST κ-ω model

Table 2: Comparison of Simulation results with the experimental data

Loading conditions	Simulation Results	Experimental results	% of error
35° guide vane opening	89.2	91.5	2.5%
40° guide vane opening	90.3	92.0	1.8%
50° guide vane opening	88.5	90.8	2.5%

5.3 Grid dependency test

For analyzing the effect of grid size on the solution obtained from CFD, grid dependency test has been performed taking four different grid sizes under steady flow condition.

From table 3 it is clear that the overall efficiency of hydraulic turbine increases as the number of nodes increases. But after the test 3 the number of nodes increases without significant increase in efficiency therefore this is our optimum number of nodes.

Table 3: Number of elements chosen for the simulation

Case No.	No. of Nodes	No. of Elements	Efficiency %
1	9,58,624	53,91,684	84.62
2	10,60,254	61,57,496	89.54
3	**11,46,459**	**63,04,220**	**90.29**
4	12,98,847	65,48,956	90.31

5.3 Effect of change of time-steps on performance of Kaplan turbine

The effect of change of time-steps to the overall performance of turbine is evidently shown in figure 2. For different guide vane openings and mass flow rates, in each case turbine with time-steps corresponding to 0.25° runner rotation (i.e. 0.00003968 sec in 35° GVO, 0.00003623 sec in 40° GVO, 0.0002380 sec in 50° GVO) gives the maximum efficiency while the turbine corresponding time-steps in 1.5° runner rotation (i.e. 0.00023809 sec in 35° GVO, 0.00021739 sec in 40° GVO, 0.00018181 sec in 50° GVO) gives the minimum efficiency. It may be because higher turbulence will be occurring at higher time-steps.

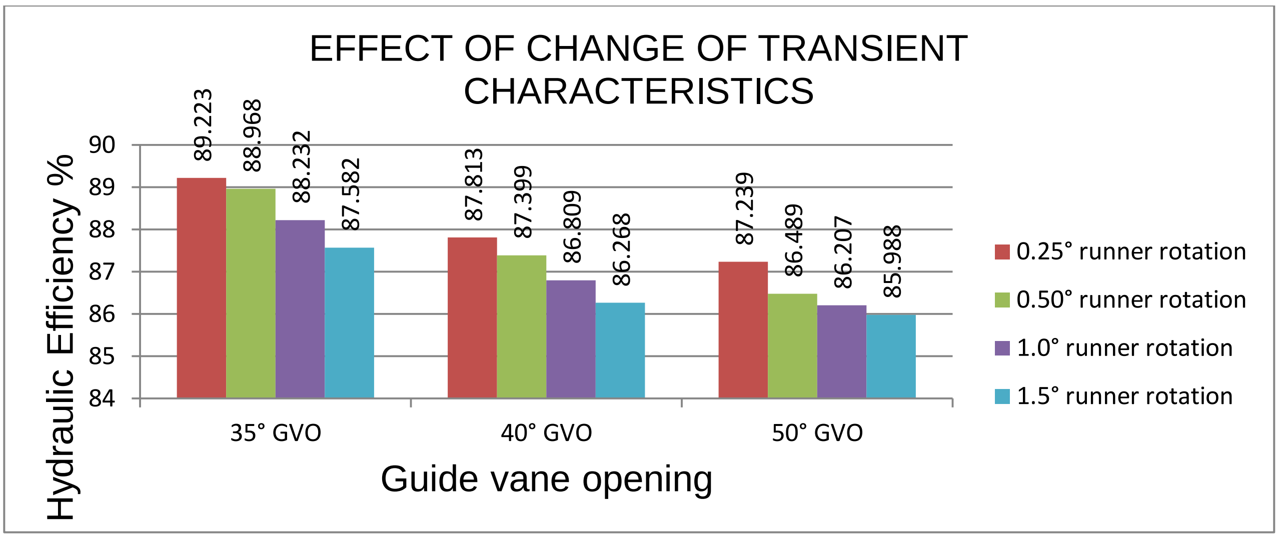

Fig. 2 - Effect of change of transient characteristics

5.4 Variation of courant number with time-steps

For a CFD simulation, the Courant number could be that which tells something about how fluid is moving through computational cells. If the Courant number is less than or equal to 1, fluid particles move from one cell to another within one time step (at most). If it is greater than 1 then fluid particles move through two or more cells at each time

ISSN 2229-5518

step and this can affect convergence negatively. The variation of courant number at inlet and outlet of turbine is shown in table 3,4 and 5.

Table 3: Variation of courant number at 35° GVO

Timesteps (runner rotation)	Courant Number at Inlet	Courant Number at Outlet
0.25°	0.0129394	0.0076569
0.50°	0.0257614	0.0152371
1.0°	0.0515229	0.0304729
1.5°	0.0776104	0.0459313

Table 4: Variation of courant number at 40° GVO

Timesteps (runner rotation)	Courant Number at Inlet	Courant Number at Outlet
0.25°	0.0139406	0.0083452
0.50°	0.0277272	0.0166036
1.0°	0.0554545	0.0332053
1.5°	0.0831817	0.0498117

Table 5: Variation of courant number at 50° GVO

Timesteps (runner rotation)	Courant Number at Inlet	Courant Number at Outlet
0.25°	0.0134377	0.0079614
0.50°	0.0268754	0.0159231
1.0°	0.0537508	0.0318458
1.5°	0.0806262	0.0477687

From the above tables it can be observed that the Courant number increases with the increase in guide vane opening at both inlet and outlet of the turbine. The value of Courant number should be less than unity otherwise there will be problem in obtaining convergence.

5.5 Variation of runner head loss with time-steps

The head across the turbine at different guide vane opening is depicted in table 6,7 and 8.

Table 6: Variation of runner head loss at 35° GVO

Runner rotation corresponding to Time-steps	Calculated runner head	Net head	Runner Head loss (%)
0.25°	9.1711	8.1828	10.7763
0.50°	9.1771	8.1648	11.0310
1.0°	9.1787	8.0986	11.7671
1.5°	9.1710	8.0322	12.4171

Table 7: Variation of runner head loss at 40° GVO

Runner rotation corresponding to Time-steps	Calculated runner head	Net head	Runner Head loss (%)
0.25°	12.6245	11.0860	12.1866
0.50°	12.6506	11.0565	12.6008
1.0°	12.6415	10.9740	13.1908
1.5°	12.6512	10.9140	13.7319

Table 8: Variation of runner head loss at 50° GVO

Runner rotation corresponding to Time-steps	Calculated runner head	Net head	Runner Head loss (%)
0.25°	13.0963	11.4251	12.7604
0.50°	13.0968	11.3273	13.5105
1.0°	13.0978	11.2912	13.7927
1.5°	13.0984	11.2631	14.0117

From above tables it is very clear that in each case runner head loss increases with the increase in timesteps and thus the percentage head loss also increases. The runner head loss are minimum at timesteps corresponding to 0.25° runner rotation and maximum at timesteps corresponding to 1.5° runner rotation in each case due to more turbulence at higher timesteps.

6. Conclusions

It may be concluded from unsteady flow simulation of Kaplan turbine that most of the flow parameters are affected by changing the time-steps in case of runner rotation i.e, from 0.25° runner rotation to 1.5° runner rotation. The following conclusions are drawn from the studies:

- It is observed that with the increase in time-steps, the efficiency decreases gradually for different operating conditions in Kaplan turbine because of increased turbulence at higher tome step.
- The Courant number increases with the increase in guide vane opening at both inlet and outlet of the turbine.
- The runner head loss also increases with the increase in time-steps.

Nomenclature

Symbol	Description
H	Net head across the turbine
H_{LR}	Rated head
H_R	Head utilized by the runner
TP_1	Total pressure at inlet
TP_2	Total pressure at outlet
ρ	Density of water
g	Gravitational acceleration
γ	Specific weight of water
P_{in}	Input power
P_{out}	Output power
Q	Mass flow rate
N	Speed of turbine
T	Torque generated in turbine
η	Hydraulic efficiency

References

- Barli V,V, Krishnamachar P. ,Desmukh M. M., Swaroop Adarsh, Gehlot V. K., “Hydraulic Turbine (Hydraulic Theory & Design Approach) Volume-II”.
- Choi, Hyen-Jun, Asid Mohammed, Hyoung-Woon, Pil-Su, Sueg-Young, Young-Ho, 2013, "CFD Validation of Performance Improvement of a 500 kW Francis Turbine", Renewable Energy, Vol.54, pp. 111-123
- Khare Ruchi, Prasad Vishnu, Mittal Sushil Kumar, 2012, "Effect of Runner Solidity on Performance of Elbow Draft Tube", Energy Procedia, Vol. 14 pp. 2054- 2059.
- Khalifa Diaelhag, 2013 “Simulation of an axial flow turbine runner’s blades using CFD” AIIC 2013, 24-26, Azores, Purtugal.
- Mulu B.G., Jonsson P.P., Cervantes M.J., 2012, "Experimental Investigation of a Kaplan Draft Tube-Part I Best Efficiency Point", Applied Energy, Vol. 93, pp.695 - 706.
- Prasad Vishnu, Gehlot V.K., Krishnamachar P., 2009 “CFD approach for design optimisation and validation for axial flow hydraulic turbine”, Indian Journal of Engineering and Material Science, pp. 229-236.
- Prasad Vishnu, Khare Ruchi, Chincholikar Abhas, 2010 , "Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations Using CFD", IIT Roorkee, India.
- Ruprecht A, Heitele M, Helmrich T., 2010 “Numerical Simulation of a Complete Francis Turbine including unsteady rotor/stator interactions”, Institute for Fluid Mechanics and Hydraulic Machinery University of Stuttgart, Germany.

Author Profile

Vaibhav Chandrakar Pursuing M-tech in Hydro Power Engineering, Department of Civil Engineering at Maulana Azad National Institute of Technology, Bhopal. Completed B.E (Mechanical Engineering) from Chhattisgarh Institute of Management and Technology, Bhilai in 2012.

Dr. Ruchi Khare completed Ph D in CFD simulation of Francis Turbine. Actively involved in model testing of hydraulic turbine and pumps . Published more than 50 papers in various national, international journals and conferences.