International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

IJIEMR Transactions, online available on 20th Sep 2021.
Link : https://ijiemr.org/downloads/Volume-10/Issue-09

Title: ANALYSIS OF AN AXIAL FLOW TURBINE FLOW WITH BLADE GEOMETRY IMPACT USING CFD

volume 10, Issue 09, Pages: 601-606

Paper Authors: **K. Chandra sekhar, B. Sudheer Prem Kumar**

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

ANALYSIS OF AN AXIAL FLOW TURBINE FLOW WITH BLADE GEOMETRY IMPACT USING CFD

K. Chandra sekhar[1] B. Sudheer Prem Kumar[2]
[1]Research Scholar, Department of Mechanical JNT University, Hyderabad, Telangana, India
Email id: sekharc333@gmail.com
[2]Professor, Department of Mechanical JNT University, Hyderabad, Telangana, India 500085
Email id: bsudheepk@jntu.ac.in

Abstract:
This study aims to use a computational fluid dynamics software package (CFX) to study and analyze flow behaviour in an axial flow turbine. This turbine is used in low head and high flow rate hydropower plants. This study indicates that the performance of the designed blades is acceptable. These blades can be used in the Kaplan turbine to produce power with some additional design modifications. These modifications can be made to reduce the blade's trailing edge vortices (in the suction side) and boundary layer forming in the blade's leading edge. Also, these modifications need further analysis and more testing using other commercial CFD in ANSYS to investigate the results. Also, this study focused on finding the variations of velocity components and the pressure by average circumferential area (ACA) from inlet to outlet of the blades and used them as factors to analyze the flow inside the blades. The results of this analysis show a good prediction of the flow behaviour inside the blades, and this leads to an acceptable blade design, which can be used in the Kaplan turbine.
Key Words: Axial flow Turbine, Blade Geometry, ANSYS, CFD.

1.0 INTRODUCTION

The aim of this study is to investigate some properties of flow field in the axial turbine stage. At first the flow in the turbine stage was modeled in the middle section of the turbine stage as the two-dimensional turbulent unsteady flow. Furthermore, a calculation of the three-dimensional unsteady inviscid flow was performed. Boundary phenomena - secondary flow - have been studied using model of the three-dimensional steady turbulent flow, using the "frozen rotor" technique. Effect of the secondary flow through the seal of the rotor was studied in a simplified model of a prismatic blade cascade using model of the three-dimensional steady turbulent flow. The turbine stage consists of the stator and rotor wheels. The stator wheel contains 70 and the rotor wheel 90 blades. Therefor it was possible consider only periodic field of seven stator and nine rotor blades for the calculation. The hub diameter of the stage is 0.506 m, the tip diameter is 0.597 m (blades span is 0.0455 m), the axial chord of the stator is 0.01955 m, the axial chord of the rotor is 0.025 m (both at the hub), the gap between stator and rotor is 0.005 m. The shape of the blades is shown in Figure 1. The angular speed of the rotor is 455.11 rad s−1. The inlet total pressure is 1.311 · 105 Pa, the inlet total temperature is 55.81◦C, the angle of attack is 0◦ and the outlet static pressure is 0.9538 · 105 Pa. The in-house URANS numerical code was developed for calculation of flow in the turbine stage.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

2.0 LITERATURE REVIEW

Practical hydraulic turbine method has become a compelling technique to capture minor details of the flow which are unbearable in the physical model testing and overcome the limitations of the physical laboratory setup [2]. Complementary to experimental investigation, the numerical simulation of flows is an auspicious way to investigate flows at real operating conditions [3]. Although the key geometric features and their effects on turbine efficiency have been experimentally studied, this knowledge does not readily help to design high-efficiency turbines, partly because of the knowledge about the details of the runner flow. As it is difficult and expensive to measure and visualize the flow fields in the runner, the alternative is computational simulation [4].)ere are various CFD simulation codes available in the industry and designed for the application of computeraided engineering.Crossflow turbines were conceived by Michell [5] who referred to them as Michell-Banki turbines or impulse turbines in which water strikes the turbine transversely across its blades.)e maximum efficiency of crossflow turbines tends to be 70–86%, which is weaker than that of more frequently used advanced turbines such as Pelton, Francis, and Kaplan, which have typical maximum efficiencies above 90% [6, 7].)e crossflow turbine operates with ambient air pressure on the free surface [8]. However, since the crossflow hydro turbines had low price and were mostly suitable for microhydropower units less than 2 MW and heads less than 200 m, they are mostly appropriate for the production of electricity particularly in rural communities of developing countries. Despite being enriched with perennial rivers, most of these communities lack electricity

3.0 METHODOLOGY

In axial flow turbine, water passes through the series of blade rows and changes its direction from redial to axial. Runner it 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 and efficiency. The rotation of the runner and operation of the turbine either below or above the rated conditions 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 its expensive and time-consuming tests. The flow in axial flow turbine (Kaplan) is very complex including several flow phenomena, such as turbulence, separation, swirling flow and unsteadiness flow. Advanced fluid flows are described by the continuity and momentum equations, which can generally not be solved analytically. Therefore, the numerical procedure in computational fluid dynamics (CFD) is of highest importance. CFD can be used to to check efficacy of alternate designs [3, 4] of turbines for optimization before final experimental testing of selected designs in resorted. However, in order to prove reliability of these tools for application to turbines, validations [5, 6] with known experimental results is requiredpresent work, 3D viscous flow simulation with SST k-ω turbulence model is carried out in

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

an experimental tested model of an axial flow hydraulic turbine using ANSYS CFX10.0 software.

Definition of Geometry:

The axial flow turbine consists of casing, stay rings, distributor, runner and draft tube. The energy transfer takes place in runner hence, present work is focused on runner blades only and therefore, analysis is carried from inlet to outlet where proper boundary condition can be applied. There are 12 stay vanes, 28 guide vanes and 6 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 flow parameters at inlet and exit of runner blade with velocity triangles are defined in Fig

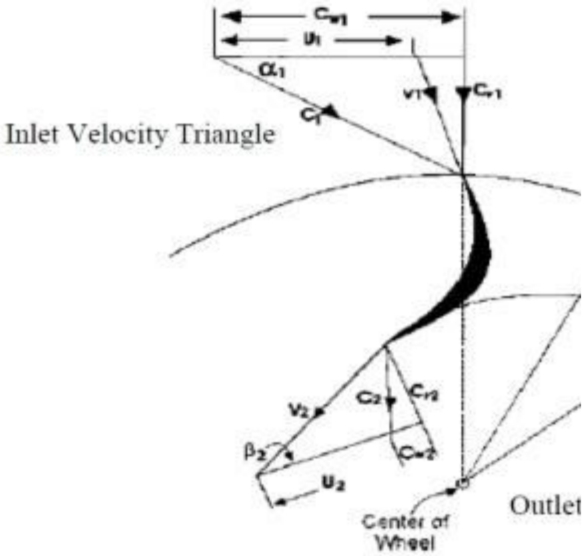


Figure: Velocity triangle at inlet and outlet of the runner blade

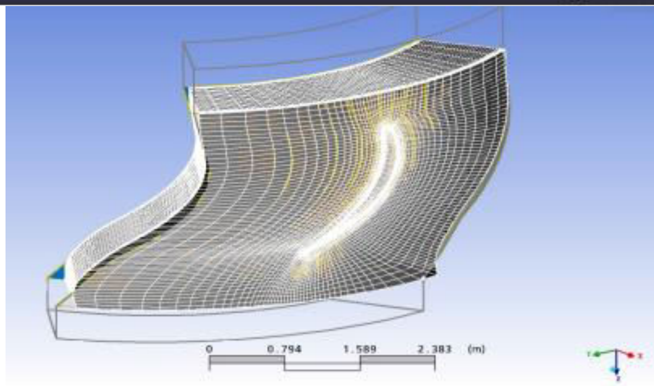


Figure: Computational domain of runner blade

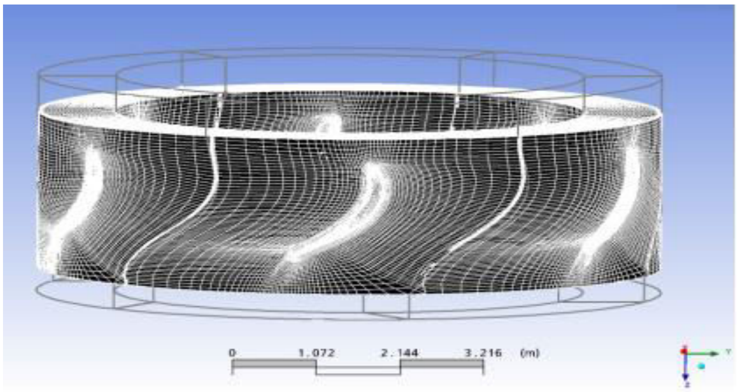


Figure: Unstructured hexahedral mesh for all domains

Boundary Condition:

The results obtained from the simulation in any flow domain depend on the specific boundary condition. In this work, the total flow rate and its direction are specified at stay vanes inlet as inlet boundary condition and static pressure 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.

Governing Equations: The flow in the runner is assumed to be turbulent, and incompressible, the Reynolds Averaged Navier Stokes (RANS) equations consisting of continuity, momentum and energy equations are used. In order to solve these governing equations in Ansys workbench, the CFX10.0 solver has been utilized in this study.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

$$\frac{\partial u_i}{\partial x_i} = 0$$

Momentum $$\rho\frac{\partial u_i}{\partial t} + \rho\frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]$$

Energy $$\rho\frac{\partial T}{\partial t} + \rho\frac{\partial T u_j}{\partial x_j} = \frac{\partial}{c_p \partial x_j}\left(K\frac{\partial T}{\partial x_j}\right)$$

Computation of Flow Parameters:

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

Pressure coefficient $C_P =$

Velocity coefficient $C_V =$

Flow deflection $\varepsilon = \beta_1 -$

Degree of reaction $\varphi = \frac{W_2{}^2}{2}$

Results and Discussion

The CFD simulations are assumed converged when all the residuals are less than10^{-7}, which is sufficient for most engineering problems. The velocity at points at the inlet, the centre and at the outlet is monitored and when there is no change in the results are considered converged. The distinct rise in the residual plot is due to the change in the differencing schemes. The convergence of the SST turbulence model in this study is assumed converged when the residuals plots drop to 10−7 and the difference of mass flow in and mass flow out is very small compared to mass flow in.

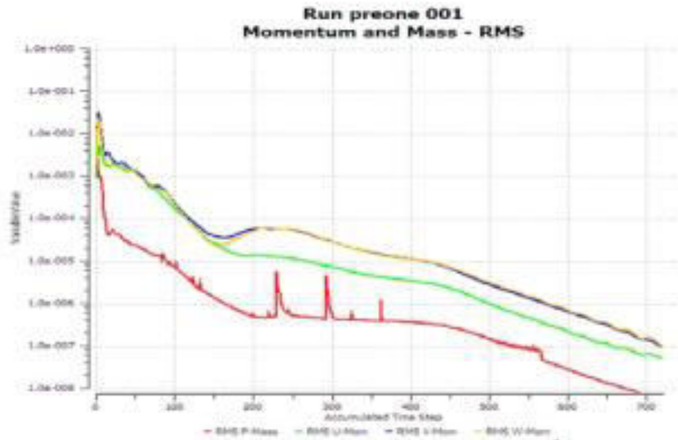
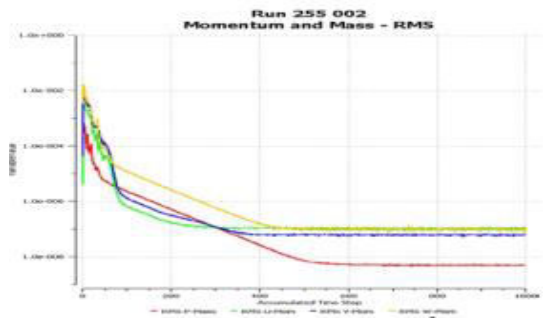


Fig (4.1) shows the residual plot of momentum and mass for three cases. From the residual of mass and momentum for three cases, (A) the case of 225*103 nodes is accepted because the residual plots are constant for long iteration and no change is the values of velocities components, (B) the case of 255*103 nodes the residual plots is less than the residual target, (C) the case of 352*103 nodes there is fluctuation in the value of v-component due to computer capabilities and grid quality, all the following results are taken for the case (B) of 255*103 nodes



Fig(4.2) show clearly the static pressure distribution between two successive blades of the turbine. Here,(it is very clear the high pressure excreted on the pressure side of blade and low pressure in blade suction side, this valid for all blades and according to the theory of turbomachinery the pressure side is higher than suction side of the blade. Results confirm that the pressure drops gradually form the inlet to outlet due

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

to the extraction of fluid energy by the turbine runner.

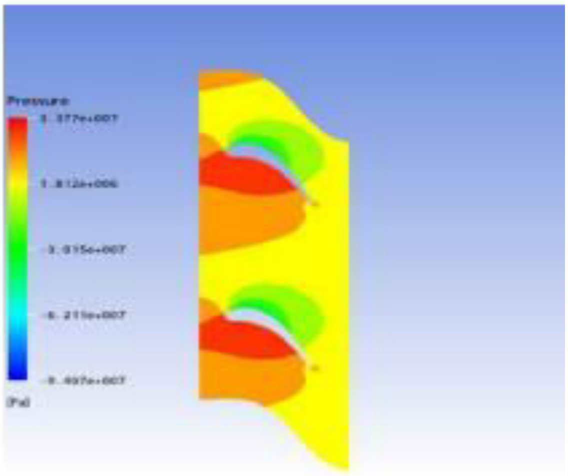


Figure 4.3: Static pressure distribution on plane at mean radius

Fig(4.3) show the pressure variation from inlet to outlet this means that the pressure decreased from inlet to leading edge and then increased at the leadig edge after that decreased gradually along the blade until reach the tailing edge then increased to leaves with atmospheric pressure.

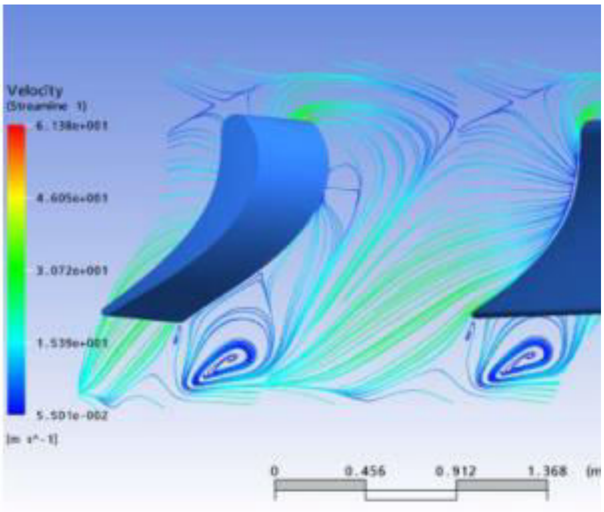


Fig (4.4) streamlines (A) on Hub surface

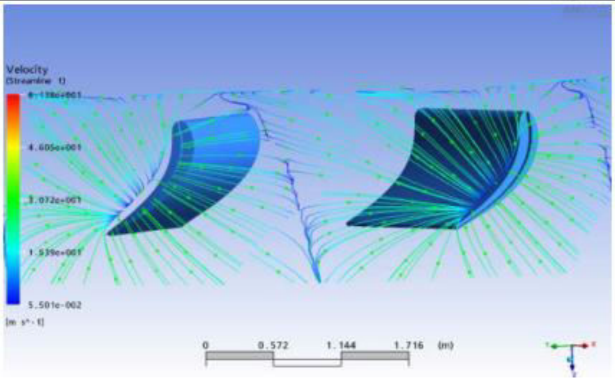


Fig (4.5) streamlines (B) on Shroud plane

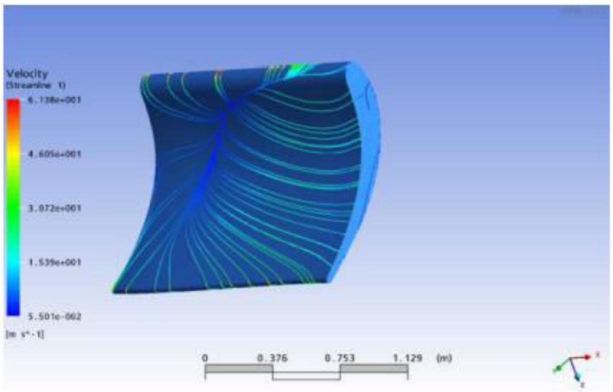


Fig (4.6) streamlines (C) On Blade-pressure surface

DISCUSSIONS

It is found from simulation results that the most of local flow parameters like velocities and flow angles at inlet and outlet are affected by the operating regimes of turbine. The variation patterns of discharge factor, efficiency and specific energy obtained from numerical simulation are well agreed with experimental results for any axial turbine. The losses are minimum at the points of maximum efficiency. The computed values of different parameters may differ from experimental one because CFD gives approximate solution of flow governing equations and accuracy depends on many factors. It can be concluded that CFD is a cost-effective computational tool for flow simulation and investigation for hydraulic turbines and can provide detailed flow information. This information will be useful in efficient design of turbine. Despite the rapid growth in the ease of use, speed and robustness of CFD tool, considerable expertise is still required to ensure accurate simulations and validation of numerical results.

Conclusions:

The numerical simulation results show similar pattern for velocity and pressure variation by average circumferential area (ACA) and the distribution between hub and shroud, efficiency and power output affected by the rotational speed of the runner. The maximum efficiency and power out but occur at the same rotational speed. The total computed loss is minimum at the point of maximum efficiency. The streamline and pressure contour plots in different component confirm with actual flow behavior 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 performances 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.

References:

[1] R. Adhikari, Design Improvement of Crossflow Hydro Turbine, University of Calgary, Calgary, Canada, 2016.

[2] S. R. Yassen, "Investigation of the effects of number of blades on the performance of cross-flow turbine using STAR CCM+," Polytechnic Journal, vol. 7, no. 4, 2017.

[3] N. Gourdain, "Large eddy simulation of flows in industrial compressors: a path from 2015 to 2035," PhilosophicalTransactions of the Royal society A: Mathematical, Physicalengineering sciences2014, vol. 372, p. 20130323, 2022. [4] R. Adhikari and D. Wood, ")e design of high efficiency crossflow hydro turbines: a review and extension," Energies, vol. 11, no. 2, p. 267, 2018.

[5] A. G. M. Michell, Impulse-Turbine, Google Patents, Alexandria, VA, USA, 1904.

[6] M. Sinagra, "Cross-Flow turbine design for variable operating conditions," Procedia Engineering, vol. 70, pp. 1539–1548, 2014.

[7] A. Elbatran, "Operation, performance and economic analysis of low head micro-hydropower turbines for rural and remote areas: a review," Renewable and Sustainable Energy Reviews, vol. 43, pp. 40–50, 2015.

[8] F. M. White and I. Corfield, Viscous Fluid Flow, McGraw-Hill, New York, NY, USA, 2006.

[9] V. Sammartano, "Numerical and experimental investigation of a cross-flow water turbine," Journal of Hydraulic Research, vol. 54, no. 3, pp. 321–331, 2016.

[10] J. De Andrade, "Numerical investigation of the internal flow in a Banki turbine," International Journal of Rotating Machinery, vol. 2011, 2011.