SIMULATION OF AN AXIAL FLOW TURBINE RUNNER'S BLADES USING CFD

Diaelhag Khalifa, M.Sc
Lecturer of Mechanical Engineering, King Abdul-Aziz University, Jeddah, KSA

Abstract:
This study aims to use computational fluid dynamics software package (CFX) to study and analyze flow's behavior in an axial flow turbine. This turbine used in low head and high flow rate hydropower plant. This study indicates that, performance of the designed blades is acceptable. These blades can be used in Kaplan turbine to produce a power with some addition design modification. These modifications can be made to reduce the tailing edge vortices (in suction side) of the blade and boundary layer forming in the leading edge of the blade. Also these modifications need furtherer analysis and more testing using another commercial CFD codes to investigate the results. Also this study focused to found the variations of velocity components and the pressure by average circumferential area (ACA) from inlet to outlet of the blades and used as factors to analyzed the flow inside the blades, the results of this analysis shows a good prediction of the flow behavior inside the blades and this lead to acceptable blade design, which can be used in Kaplan turbine.

Key Words: Designing, Simulation, CFD, Numerical, ANSYS, CFX10.0, Axial turbines, Verification, Visualization

Introduction

Hydro power plants generate one fifth of the total electrical power produced in world [1]. Even a small improvement of the hydrodynamic design and efficiency can contribute a great deal to the supply of the electric power. The efficiency of a hydropower plant depends on a number of parameters, such as: Turbine efficiency, Draft tube efficiency and Generator efficiency [2]. Most of the past studies have focused on the draft tube for increasing the efficiency of the plant, but a good draft tube design is not enough. Recent studies have shown that the efficiency improvement can also be realized by minor modification on the older design in the rest of the waterway i.e., in the draft tube, runner blades and spiral casing. Previous studies have shown that there is potential for increasing unit performance by a moderate modification of such runner blades. Runner blades have been found to have an efficiency loss due to the runner losses. A small increase in performance in these power stations represents a considerable economic value. This work will analyzed the flow field in the runner blades. This analysis is based on CFD simulations.

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 required. In present work, 3D viscous flow simulation with SST k-ω turbulence model is carried out in an experimental tested model of an axial flow hydraulic turbine using ANSYS CFX10.0 software. The variation of flow parameters from hub to tip of runner are presented in graphical form and average value of cascade parameters are computed at diffener operating regimes.

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.1. The computational domain of runner blade is shown in Fig. 2, and the unstructured hexahedral mesh is generated in ANSYS workbench for all domains are shown in Fig. 3. The y^+ varies between 24 to 186, which is the acceptable rage for automatic wall function treatment in boundary layer in SST k-ω turbulence model [7]. The maximum values other mesh quality parameters like face angle, edge length ratio, connectivity number are within acceptable limits of Ansys CFX10.0.

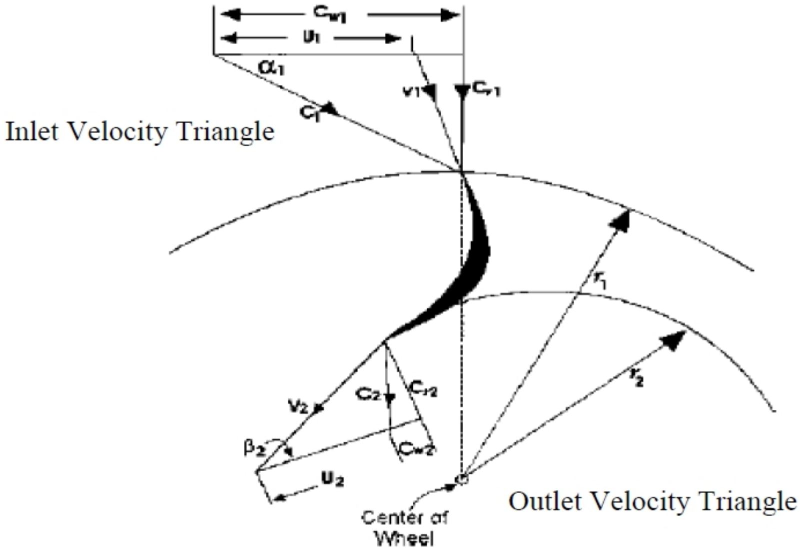


Fig. 1 – Velocity triangle at inlet and outlet of the runner blade

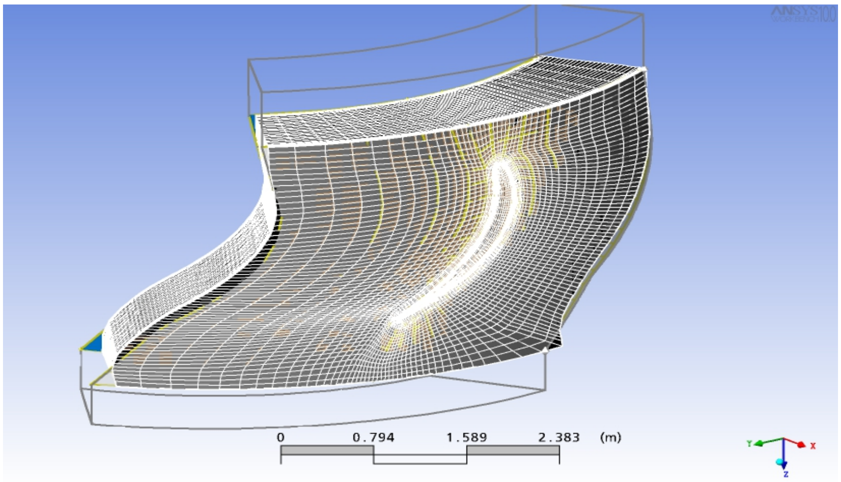


Fig. 2- Computational domain of runner blade

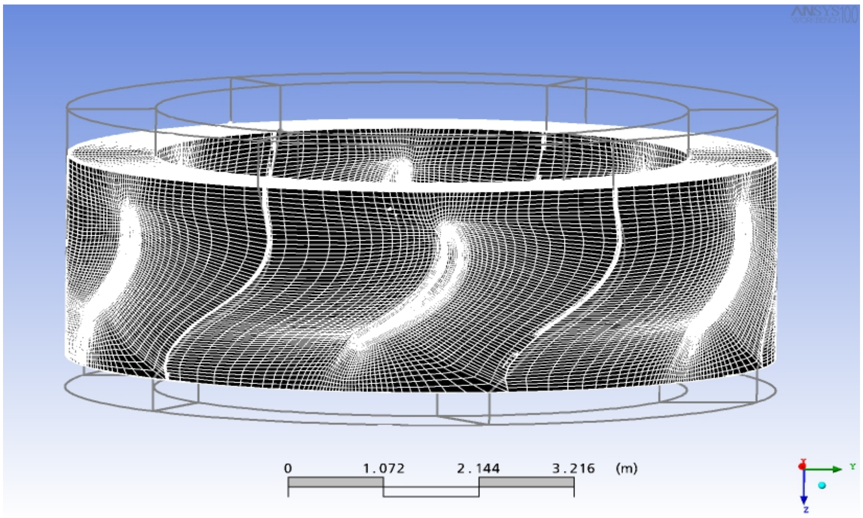


Fig. 3-Unstructured hexahedral mesh for all domains

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.

Continuity $$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Momentum $$\rho\frac{\partial u_i}{\partial t} + \rho\frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

Energy $$\rho\frac{\partial T}{\partial t} + \rho\frac{\partial T u_j}{\partial x_j} = \frac{\partial}{c_p \partial x_j}\left(\mathrm{K}\frac{\partial T}{\partial x_j}\right) \tag{3}$$

These equations are non-linear and their analytical solution is not possible and hence approximate solution is obtained by numerical method [8] .

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 [9]:

Pressure coefficient $$C_P = \frac{P - P_2}{\frac{1}{2}\rho {W_2}^2} \tag{4}$$

Velocity coefficient $$C_V = \frac{W}{W_2} \tag{5}$$

Flow deflection $$\varepsilon = \beta_1 - \beta_2 \tag{6}$$

Degree of reaction $$\varphi = \frac{{W_2}^2 - {W_1}^2}{2gH} \tag{7}$$

Circulation coefficient $$\tau = \frac{t(C_{U1} - C_{U2})}{D\sqrt{2gH}} \tag{8}$$

Lift coefficient $$C_L = 2\frac{t}{l}\sin\beta_m(\cot\beta_2 - \cot\beta_1) \tag{9}$$

Runner energy coefficient $$\emptyset = \frac{gH_R D^4}{Q^2} \tag{10}$$

Total Energy coefficient $$\psi = \frac{gHD^4}{Q^2} \tag{11}$$

Total head $$H = \frac{TP_{SV1} - TP_{DET}}{\gamma} \tag{12}$$

Head utilized by runner $H_R = \frac{TP_1 - TP_2}{\gamma} - H_{FR}$ (13)

Hydraulic efficiency $\eta_H = \frac{H_R}{H} * 100$ (14)

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 (5.1) 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 $225*10^3$ nodes is accepted because the residual plots is constant for long iteration and no change is the values of velocities components, (B) the case of $255*10^3$ nodes the residual plots is less than the residual target, (C) the case of $352*10^3$ 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 $255*10^3$ nodes.

Fig(5.2) show clearly the static pressure distribution between two successive blades of the turbine. Here,(it is very clear the high pressure excerted on the pressure side of blade and low pressure in blade suction side, this valid for all blades and according to the theorm of terbomachinery the pressure side is higher than suction side of the blade.

Results confirm that the pressure drops gradually form the inlet to outlet due to the extraction of fluid energy by the turbine runner.

Fig(5.3) 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 atomspheric pressure.

Fig(5.4) show the variation velocity component from inlet to outlet along the streamwise,the velocity component U in x-direction starting from small value near to zero and incearesed gradually along the streamwise(radial direction of blade), the velocity component V in Y- direction is very small startnig from negative value and increased gradually along the streamwise until reach the vlaue less than zero this in the direction of blade span,the velocity component W in Z-direction is semi constant along the streamwise this in the direction of rotational but the total velocity decreased from inlet to outlet due to rotation, swirling and vortex occur.

Fig(5.5) show the performance curves of the of blade under different operation condition, this indicate that the efficiency of the turbine balde increased with increasing of N rpm until reach maximum poit and then decreased gradully (parabolic shap) this valid only for N from 120 to 150 rpm out of this range the efficiency is varies.The power also increased with the increase of N rpm to maximum also this is valid for N from 120 to 160 rpm.

Fig (5.6) show the streamlines at hub,shroud and blade, (A) at the hub this indicate that, there is vrotex occur at the suction side of the blade near the tailing edge where the low pressure region.(B) at the shroud this means that the water flow path in the tip clearance. At (C) the streamlines in pressure surface of blade indicate the path of water flow over pressure side, (D) the streamlines at suction surface of the blade where the vortexes occur and it's very clear at velocity vector.

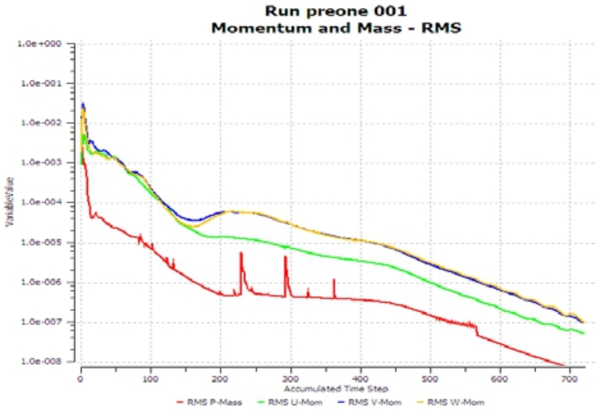


Fig. 5.1 Residual plots for (A) 225*10^3 nodes

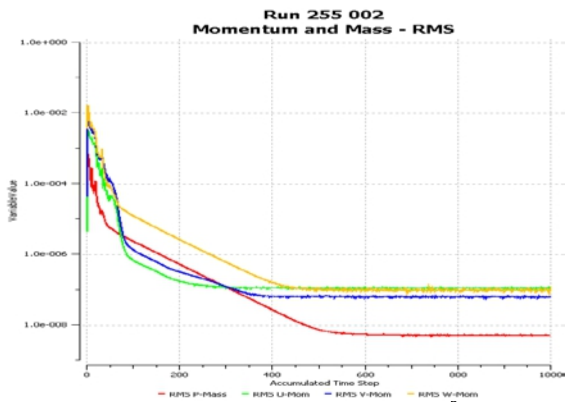


Fig.5.1 Residual plots for (B) 255*10^3 nodes

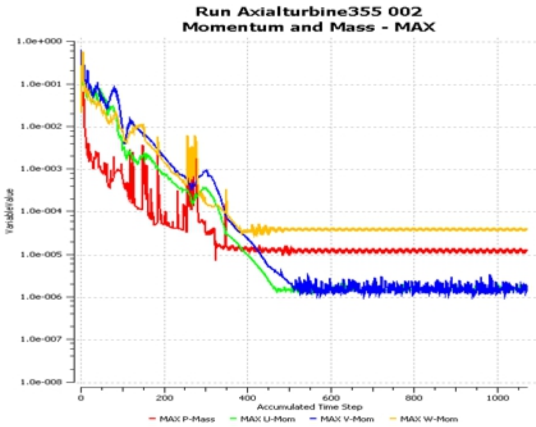


Fig: 5.1 Residual plots for (C) 352*10^3 nodes

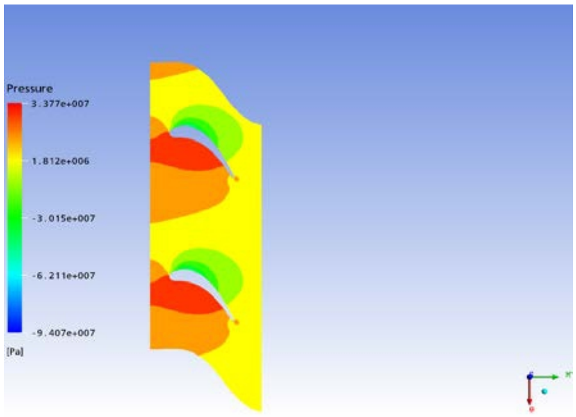


Fig: 5.2 Static pressure distribution on plane at mean radius

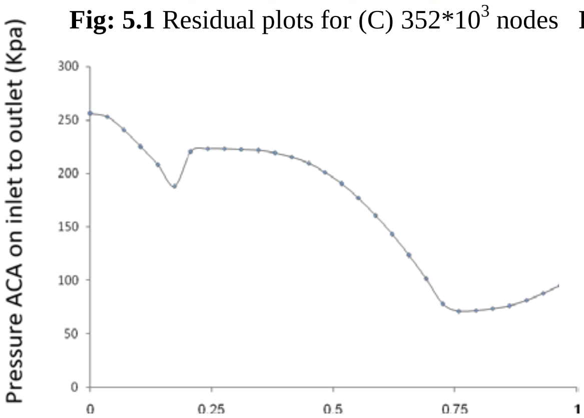


Fig: 5.3 Variation of Pressure ACA form inlet to outlet

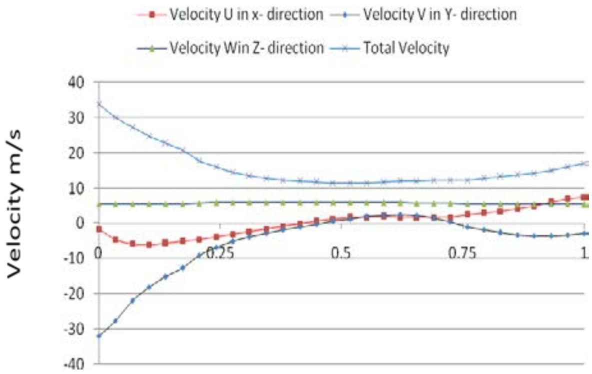
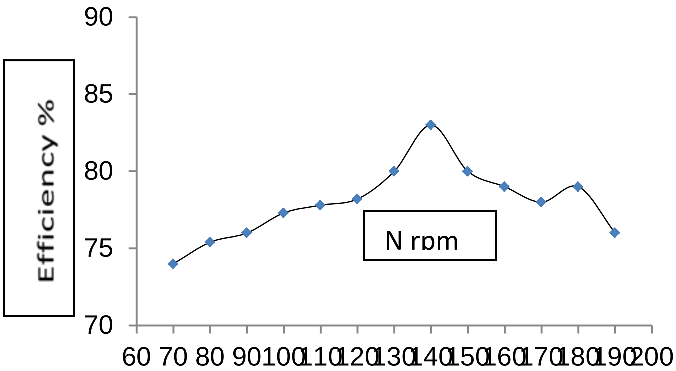


Fig: 5.4 Variation of velocities from inlet to outlet



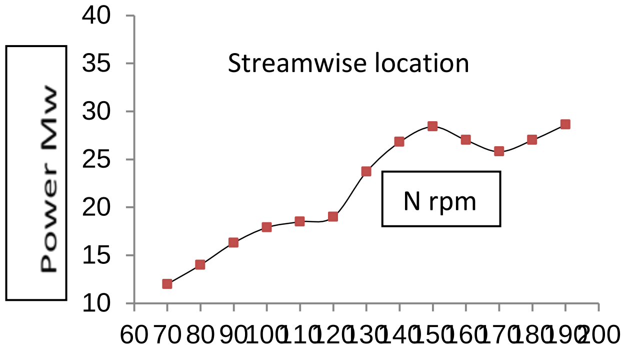


Fig (5.5) Performance Curves Efficiency and Power vs Rotational Speed

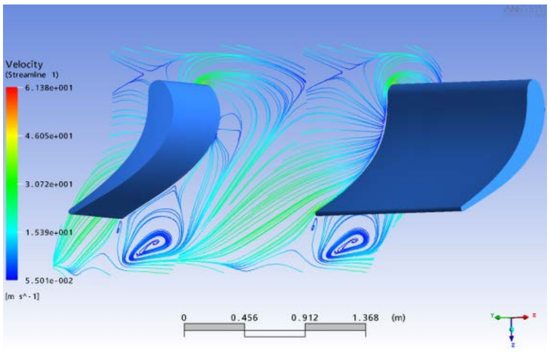


Fig: 5.6 (A)

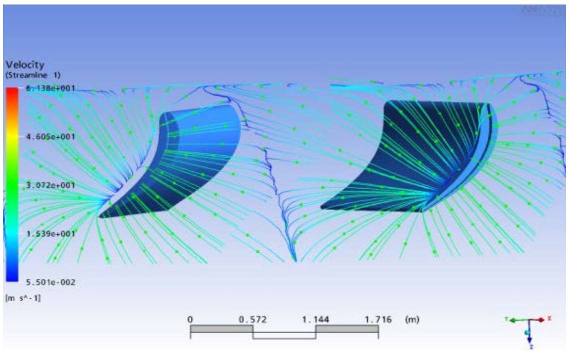


Fig: 5.6 (B)

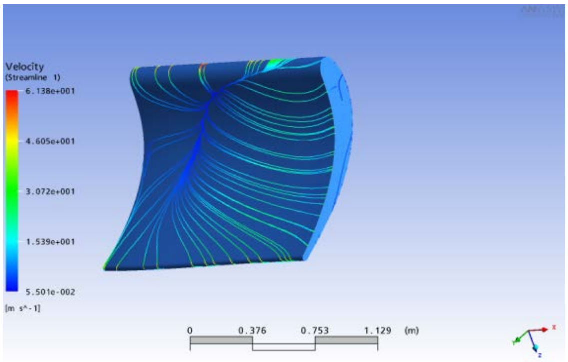


Fig: 5.6 (C)

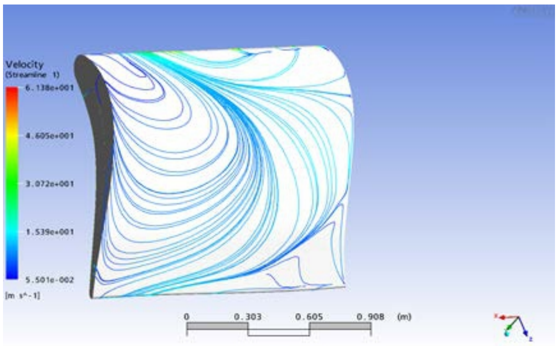


Fig: 5.6 (D)

Fig (5.6) streamlines (A) on Hub surface (B) on Shroud plane (C) On Blade-pressure surface (D) on Blade-suction surface.

Conclusion

The numerical simulation results shows 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 occurs 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 indentified 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.

Nomenclature

$C =$ absolute flow velocity (m/s)
C_U = whirl velocity (m/s)
D = diameter of turbine runner (m)
G= gravitational acceleration (m/s^2)
H = net head (m)
H_{DTR} = head recovery in draft tube (m)
H_{FR} = head loss in runner (m)
L = blade chord (m)
P = static pressure at any point on blade surface profile (Pa)
Q = discharge through turbine (m^3/s)
TP = total pressure at runner (N/m^2)
TP_{SV1} = total pressure at stay vane inlet (N/m^2)
TP_{DTE} = total pressure at draft tube outlet (N/m^2)
T = Pitch of runner blades(m)

W = relative velocity at any point of blade surface (m/s)
α = guide vane angle from tangential direction (^{0})
β = relative flow angle (^{0})
β_m = mean relative flow angle (^{0})
γ = specific weight of water (N/m^3)
ρ = mass density of water (kg/m^3)
subscript 1 and 2 denote values of parameters at inlet and outlet of runner respectively

References :

1- World Energy Council, "*Survey of Energy Resources*", Technical Report, Sep.2006.
2- Krivchenko G., "*Hydraulic Machines: Turbines and Pumps*",2^{ed}., Lewis, Boca Raton. 1994.
3- Guoyi P, J ''*Fluid Engineering*'', 27 (2005)1183-1190
4- Wu J, Shimmel K, Tani K, Niikura K & Sato J, '' *Fluid Engineering*'',127 (2007) 159- 168.
5- Shukla, M, *CFD analysis of 3-D Flow and its validation for francis turbine*, M.Tch. Thesis, Maulana Azad National Institute of Technology, Bhopal, 2007
6- Rao V S & Tripathi S K , ''*Pro Nat Seminar on CFD-The 3rd Dimension in Flow analysis & Thermal Design*'', Bhopal, India, (2007) 196-201.
7- ANSYS, ''*Ansys CFX10.0 Help Manual*'', Bangalore: ANSYS Software Pvt. Ltd, 2005.
8- Lewis, R.I., ''*Turbo Machinery Performance Analysis* ''(Arnold London), 1996.
9- Raabe I J, ''*Hydro Power- The Design, use and Function of hydro mechanical hydraulic and electrical equipment*'' (VDI-Verlag, GmbH, Dusseldorf), 1985.