CFD Approach for Flow Investigations in Hydraulic Reaction Turbines – A Review

Sanjay Jain[1], R. P. Saini[2], and Arun Kumar[3]
[1]Nirma University of Science and Technology, Ahmedabad-382481
[2]Indian Institute of Technology Roorkee, Roorkee, Uttarakhand-247667
[3]Indian Institute of Technology Roorkee, Roorkee, Uttarakhand-247667
Corresponding Author: svjain5@yahoo.com

***Abstract*— For cost-effective design of hydro turbines it is essential to design a system according to fluid behavior in the flow passage. Due to complexity of the flow many times it may not be practically feasible to carry out experimental investigations. Instead, flow can be simulated numerically comparatively easily and at low cost using Computational Fluid Dynamics (CFD) approach. Many researchers have used CFD technique for flow simulation in hydraulic turbomachinery. This paper reviews the research being carried out by different investigators in the hydraulic Reaction turbines by CFD approach. Starting with Q3D-Euler and 3D-Euler, today usually Reynolds averaged Navier-Stokes equations together with robust models of turbulence like standard k-ε is used. The current industry standard for CFD simulation in hydraulic turbomachinery is also presented.**

***Index Terms*— Computational Fluid Dynamics, Hydraulic turbine, Reaction turbine, Review.**

I. Introduction

THE world is facing twin energy-related threats: that of not having adequate and secure supplies of energy at affordable prices and that of environmental harm caused by consuming too much of it. Rising energy prices and recent geopolitical events have reminded us of the essential role, energy plays in economic growth and human development activities.

According to International Energy Agency (IEA), about 1000 GW of existing thermal electricity plants in the world will be decommissioned by 2030 [1]. Also, realizing the Kyoto protocol and Brazil summits held to reduce the green house gas emissions, hydropower has became one of the top power development option to meet the increasing energy demand.

The harnessing of energy of the falling water to provide mechanical power has been one of man's greatest achievements. Water wheels providing mechanical power for grinding and water pumping are still in common use. The industrial revolution created new requirements, which demanded large hydro power plants which led to rapid improvement in the design of hydro turbines and water conductor systems in hydropower sector.

Turbines are designed according to specific site parameters like head and discharge. Hence, for cost-effective design of hydro components it is essential to design a system according to fluid behavior in the flow passage. The flow in hydraulic components is highly complex due to three dimensional nature, turbulence and viscous effects. It is very difficult, time consuming and costly to perform experiments either on models or on prototypes. Instead, flow can be simulated numerically relatively easily and at low cost using commercially available CFD packages like Fluent, CFX, Star-CD etc.

Computational Fluid Dynamics (CFD) is the tool for the numerical simulation of flow in any fluid system. Its a powerful technique that provides an approximate solution numerically to the coupled governing fluid flow equations for mass, momentum and energy transport. The flexibility of the technique makes it possible to solve these equations in very complex spaces, unlike simpler modeling methods that are sometimes used for simpler designs. In CFD, the flow domain is descretized into number of cells or elements forming a grid. Then fundamental equations governing the flow are solved at different nodes till desired convergence is achieved.

The study of flow field in the hydropower plants can be broadly divided into two categories. The first-one deals with water conductor system which includes intake channel, penstock, surge tank, spillway, tailrace channel etc. and the second deals with hydro turbine which is obviously most critical component. Many researchers have applied CFD as a tool for flow simulation in different components of hydropower plants. Starting with simplified methods based on two dimensional potential theory, which did not take into account frictional effects, nowadays CFD methods include three-dimensional, viscous and turbulence effects, rotor-stator interaction, multiphase flow, transient effects in the flow etc [2].

It is the purpose of this review to examine the different investigations being carried out by different investigators in the Reaction turbines, which is generally suitable for medium to low head hydropower plants, by CFD approach. A CFD approach for specific problems related to different reaction turbines is discussed. The current industry standard for CFD simulation in hydraulic turbomachinery is also presented.

II. CFD FOR WATER TURBINES

Water turbines are divided principally in two groups; impulse turbines and reaction turbines. The precise shape of water turbine blades is a function of the supply pressure of water, and the type of impeller selected. In Impulse turbines, the pressure energy of water is converted into kinetic energy in nozzle and then water jet strikes the turbine blade with high velocity. They are suitable for high head applications. Examples of Impulse turbine are: Pelton turbine, Turgo-Impulse turbine, Cross flow turbine etc. Reaction turbines are acted on by water, which changes pressure as it moves through the turbine and gives up its energy. They must be encased to contain the water pressure, or they must be fully submerged in the water flow. They are used in low and medium head applications. Examples of Reaction turbine are: Francis turbine, Kaplan turbine, Bulb turbine etc.

Flow analysis in hydro power generation is always being an exhaustive subject of exploration. In practice, many times it may not be cost-effective to carry out experimental investigations. CFD approach can be used as a tool for numerical simulation of flow in turbomachinery. Many researchers have used CFD technique for flow simulation in hydraulic turbomachinery. Starting with Q3D-Euler and 3D-Euler, today usually Reynolds averaged Navier-Stokes equations together with robust models of turbulence like standard k-ε is used [3]. The current industry trend for flow simulation in hydraulic turbomachinery is given in Table I [2].

TABLE I
TODAY'S INDUSTRY STANDARD FOR CFD SIMULATION

Equation	3D Navier Stokes
Model	Viscous, turbulent
Application	Feasibility, optimization
Computational domain	Spiral casing with stay vanes Spiral casing – stator
Computational grid	Structured multiblock or unstructured, 500k-1000k elements (including stay vanes)
Discretization scheme	First to second order accuracy
Turbulence model	Two equation models: k-ε, k-ω
Time dependence	Steady

Michel *et al.* [4] mentioned from their experience that, in case of Reaction turbines, in most of the cases runner and guide vanes can be easily replaced, but modifications affecting the mechanical structure of the machine such as stay vanes reshaping or the concreted parts such as spiral case or draft tubes modifications were rare and only happened in extreme cases. They felt that, to achieve the better performance, deep study in the form of CFD calculations and then a checking by model test is generally necessary.

III. CFD APPROACH FOR MIXED FLOW TURBINES

In Francis turbine, water flows in radially inward direction and comes out in axial direction i.e. mixed flow. Different researchers have carried out analysis for the cost effective design of Francis turbine, using numerical analysis tool CFD. A team from Sulzer Hydro and Sulzer Innotec. [5] modelled a complete Francis turbine-*from the inlet of the spiral casing to the draft tube outlet*- using a 3D Navier Stokes code which can be used to design new runners that match existing components more accurately, at a lower cost than by using model tests.

Huang [6] analyzed important aspects of turbine flow characteristics such as anti-gravitational flow, spiral flow, helical flow, energy conversion in the turbine runner, meridional flow, inflow velocity moment distribution and variation of velocity moment in the turbine runner. Based on above analysis, he proposed a new design of turbine which improves turbine mechanical performance, minimizes fish injury and mortality in the turbine flow passage and at the same time, reduces the costs and time of hydropower development.

One of the most distinguishing features of today's deregulated power market is its variable power demand. This is very challenging task for turbine designer as the profitability often depends on part load performance of the turbine. This problem is more severe in Francis turbine compared to Kaplan turbine as the runner blades of Francis turbine are fixed unlike in Kaplan turbine. Many researchers have applied CFD approach to study the part load behavior of Francis turbine.

Ciocan et al. [7] presented a CFD methodology to study the unsteady rotating vortex in the draft tube of a Francis turbine at part load conditions and associated experimental study of the flow phenomena. They performed unsteady Reynolds-Averaged Navier-Stokes (URANS) simulation for the flow and validated the same with experimental results. Computation flow domain and mesh for different configurations is shown in Fig. 1. They carried out experimental investigations for a range of Thoma cavitation numbers varying from σ =1.18, *cavitation free conditions*, to σ = 0.38, *maximum rope volume*.

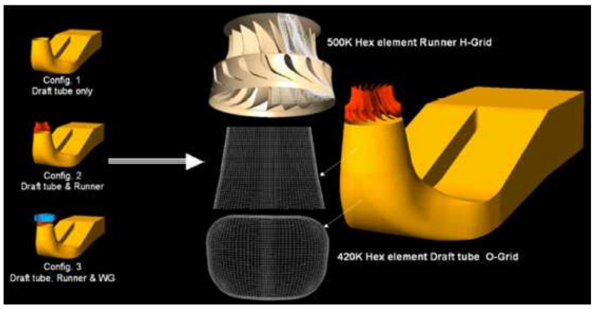

Fig. 1. Computation flow domain & mesh for different configurations

Jain *et al.* [8] carried out the flow analysis of Francis turbine at design as well as off-design conditions. They used three different turbulence models namely; standard k-ε, Renormalization group (RNG) k-ε and k-ω shear stress transport (SST) models. They found that k-ω SST model is better suited for simulation in hydraulic machinery. They observed low pressure zone near the inlet potion of the draft tube at part load operating conditions. The grid for the whole computational domain and the efficiency versus discharge curve are shown in Figs. 2 and 3 respectively.

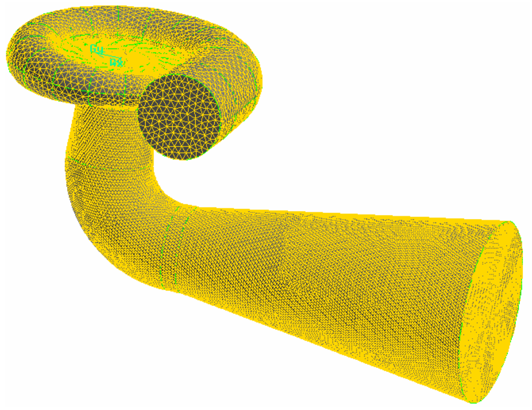

Fig. 2. Grid for the computational domain

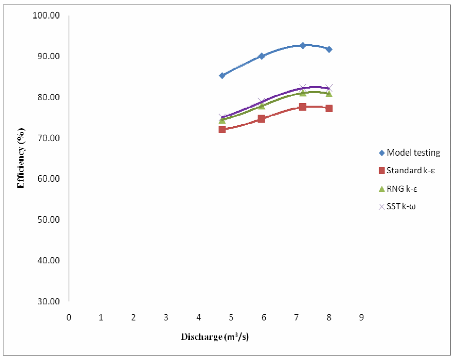

Fig. 3. Efficiency versus discharge curve for different operating conditions.

Wang and Zhou [9] measured pressure oscillations caused by vortex rope in the draft tube of a prototype Francis turbine. The three-dimensional, unsteady Reynolds-averaged Navier-Stokes equations with the RNG k-ε turbulence model were solved to model the flow within the entire flow path of the prototype hydraulic unit including the guide vanes, the runner, and the draft tube and validated the same with experiments. They found that, the amplitudes of the pressure oscillations on the blades at the lower load conditions (67% opening) were higher than at higher load conditions (83% opening) and fluctuations on the suction side tended to be stronger than on the pressure side.

Keck *et al.* [10] developed a simulation tools with which the three-dimensional turbulent flow in a complete water turbine can be simulated. Even operating points far from the best point can be calculated with astonishing accuracy, so that complete turbine hill-chart can be generated by computer. The simulated results were in excellent agreement with measurements in turbine model tests.

Few investigators have suggested some techniques to improve the operational performance at partial discharge. Jain *et al.* [8] applied two different techniques to improve the turbine operational performance at part load operating conditions and found that non uniformities can be reduced in the draft tube by introducing vane in the bend portion of the draft tube. However, they felt that the techniques need to be further optimized in terms of vane geometry, its dimensions, location etc.

Resiga *et al.* [11] found that the flow stability characteristics change when decreasing the discharge in Francis turbine. They showed that the swirling flow in the downstream of the runner, in the draft tube cone, reaches a critical state in the neighborhood of the best efficiency operating point. For larger discharge, the swirling flow is supercritical, and thus it is not able to sustain axi-symmetrical perturbations. However, at partial discharge the flow becomes subcritical and it is able to sustain axi-symmetric perturbations. They applied the jet from the runner, axially at the draft tube inlet and found that it effectively suppresses the vortex breakdown. However, if the jet velocity is too large the flow detaches from the cone wall. They found that, the required jet discharge reaches as high as 10% of the incoming discharge. As a result, in order to efficiently apply the jet control technique to hydraulic turbines, they proposed a flow-feedback approach which uses a fraction of the discharge taken through an annular slit at the end of the diffuser cone, and directs it to the jet nozzle, as shown in Fig. 4. They simulated the conical diffuser and numerically showed that this flow feedback can remove the vortex breakdown and significantly improve the pressure recovery in the conical diffuser.

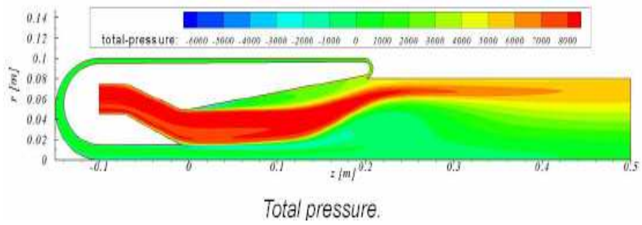

Fig. 4. Swirling flow apparatus with feedback flow control

Due to variable operational demand according to power requirement, hydrodynamic effects in turbines such as draft tube rope at part load and the guide vane-runner blade interaction become important issue in the design of these machines. The pressure waves generated by the interaction between individual blades and guide vanes propagate circumferentially in the radial spaces which results into strong machine vibrations. Hence, it is necessary to study the interaction between the different components of the turbine. Many investigators have simulated the flow around the stationary and rotating parts of the turbine.

Wu et al. [12] presented CFD-based design system for Francis turbine, which integrates three blade design approaches, parameterized geometry models, automatic mesh generators, and CFD software. They conducted extensive turbulent flow simulations for both the existing and new designs at the corresponding optimum and off design conditions. To analyze the interactions between the runner and the guide vanes, they carried out coupled calculations based on the implicit coupling method. They verified the results by model testing and achieved thoroughly improved cavitation characteristics and optimized runner and guide vanes which ultimately results into extremely smooth performance over a much wider range of operation. The optimized guide vane and stay vane profiles are shown in Fig. 5.

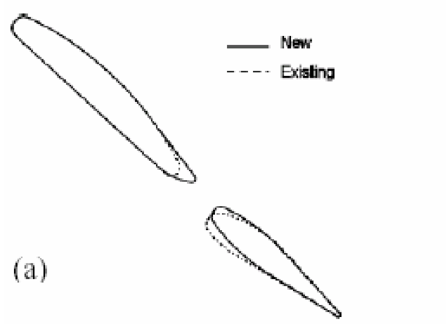

Fig. 5. Optimized guide vane and stay vane profiles.

Sick et al. [13] described the application of `stage capability' in CFD for industrial turbomachinery flow simulations. They presented the method of coupling stationary and rotating components together in one simultaneous steady-state calculation, for complete turbomachines and different applications were shown. For a Francis turbine of high specific speed, details of the CFD results are discussed and compared to experimental data which shows very good agreement between the numerical prediction and the experimental data. They also discussed the advantages of coupled stage simulations for turbomachinery design and analysis.

Sebestyen et al. [14] modified the high-pressure part (i.e., the distributor) of Francis turbine which led to optimization of the hydraulic performance of the stay-vane profiles as well as improved efficiency.

Design, operation and refurbishment of reaction turbines are strongly related to the cavitation phenomena, which may occur in rotating or stationary parts of the machine. Also, the operating range of turbine is strongly related to cavitation. Many investigators have analyzed the cavitation in the hydro turbines. Bajic [15] introduced a novel technique for diagnostics of turbine cavitation in a Francis turbine which enables identification of different cavitation mechanisms functioning in a turbine and delivers detailed turbine cavitation characteristics, for each of the mechanisms or for the total cavitation. He conducted the experiments and reported that how a turbine cavitation behavior can be improved and how a turbine operation can be optimized with respect to cavitation erosion.

The other kind of mixed flow turbine is Pump as Turbine (PAT) which is generally suitable for micro hydropower and gaining importance due to low cost and ready availability in the market. In micro hydropower, as the power generation capacity is less generally cost is equally important criteria besides efficiency. In PAT, the direction of water is changed and hence pump will work as turbine. This technology is still under development stages. In micro hydropower, the use of standard PAT may often be an alternative with a considerable economic advantage and might therefore contribute to a broader application. The only difficulty is that a PAT cannot make use of the available water as efficiently as a turbine due to its lack of hydraulic controls. Efficiencies of pumps used as turbines may be the same as in pump mode but are more often 3 - 5% lower. Many researchers have carried out flow simulation of pump in turbine mode.

Cooper *et al.* [16] explained the feasibility of using large vertical pumps for small scale Hydropower. They discussed the concept of pump as turbine, types of pumps, hydraulics of pump, characteristics curves for pumps in reverse mode, economic assessment etc.

A. Tamm *et al.* [17] carried out CFD simulations of a spiral casing test pump having a specific speed of N_S of 620. The investigation in a turbine mode has been carried out for three rotational speeds with the aim to determine the head and flow rate corresponding to the optimum efficiency. The characteristic curves at different speeds in different modes of operation are shown in Figs. 6 and Fig. 7.

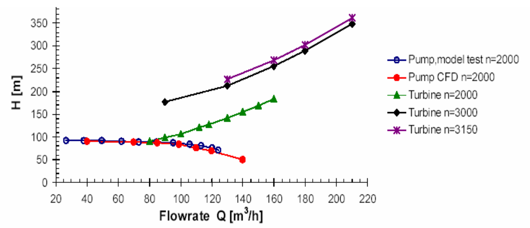

Fig. 6. Head (H) vs. Flow Rate (Q) for pumping and turbine mode

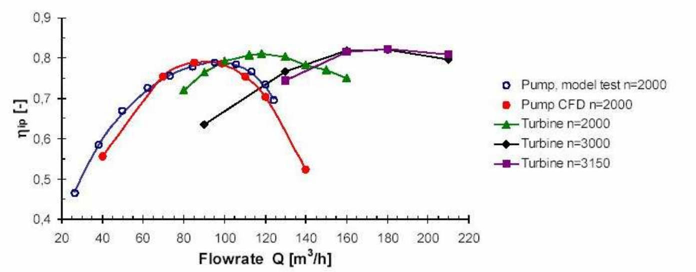

Fig. 7. Efficiency (η_{lp}) vs. Flow Rate (Q) for pumping and turbine mode

A. Shukla *et al.* [18] used CFD for simulation of flow for pump in turbine mode. They applied different turbulence models at different operating positions. They optimized the selection parameters for pump in turbine mode and found that pump can be used in turbine mode with little fall in efficiency.

IV. CFD APPROACH FOR AXIAL FLOW TURBINES

In low head applications, for the same power generation more discharge is required compared to Francis turbine. It is very difficult to accommodate huge discharge in mixed flow turbines and equally difficult to change the direction flow of water from radial to axial direction. In such cases, axial flow reaction turbines such as Kaplan, Propeller and Bulb turbines are used. The CFD approach for the flow simulation is quite similar to that of mixed flow turbine except that the flow direction is purely axial and also the blade profile is different.

The team of U S Army Corps of Engineers [19] evaluated potential environmental and performance gains that can be achieved in a Kaplan turbine through non-structural modifications to stay vane and wicket gate assemblies. After thorough analysis, they found that minor changes in the profile and configuration of the stay vane and wicket gate appear to result in improved hydro performance in terms of reduction of losses and increase in efficiency as well as improved quality of flow for fish passage.

Eisinger and Ruprecht [20] applied automatic shape optimization procedure based on mathematical optimization algorithms to optimize the shape of turbine draft tube of Kaplan turbine. This technique has been already used in the field of aerodynamics since many years for the improvement of the airfoil shapes. They presented several optimization algorithms and found that the EXTREM algorithm has proved to be the fastest in terms of optimization runs. However, a decisive drawback is that it can lead to detection of a local minimum instead of the global minimum. Nevertheless, this problem can be solved with good initial guesses. On the other hand the genetic algorithm shows a very robust behavior, but it requires a distinctly more optimization cycles and computational time.

Avellan [21] presented the cavitation phenomena featured by Francis and Kaplan turbines, including type of cavity development related to the specific speed of machine considering the influence of the operating conditions like load, head and submergence. He discussed the influence of cavitation development on machine efficiency, operation and integrity. He emphasized the importance of model testing for the proper level setting of the turbine

Hydroelectric power plants and the associated dams provide a broad range of benefits e.g., electric power, flood control, navigation, recreation, and water supply but may have some potentially negative environmental impacts also e.g., on water quality and fisheries. It is observed that the fish survival rate in Francis turbines is around 60 to 90%, *depending on the fish size and runner speed*, and that in Kaplan turbines is around 85 to 96%.

George E. Hecker et al. [22] developed a new design of Kaplan turbine to improve fish survival during their downstream migration which includes a normal scroll case, few but relatively long wicket gates, a streamlined downturn before the runner inlet, a runner with only three helical blades attached to a tapered hub and a rotating shroud attached to the outside edges of the blades. Fig. 8 shows Kaplan turbine with three helical blades. They conducted a model test and with some modifications implemented the results to the full scale turbine, and found that the full-scale turbine would have an efficiency of about 90% and that fish survival would be about 98%.

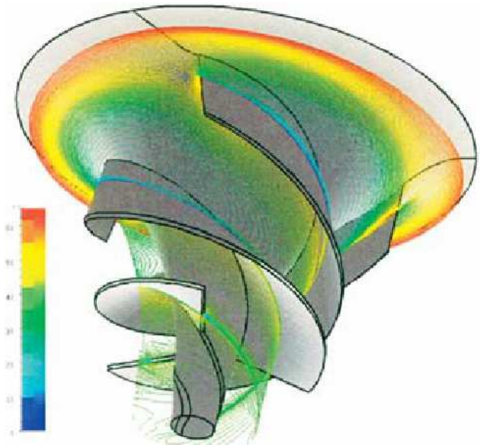

Fig. 8. Fish-friendly Kaplan turbine.

In very low head ranges, the submergence of turbine increases which leads to higher excavation cost. In such cases, Bulb turbines are used as in this case the shaft can be kept in inclined or horizontal positions. Many researchers have used CFD approach to simulate the flow in pump in turbine mode. Pan [23] developed CFD solver software to predict the performance of the low head bulb turbine. He simulated the flow through a complete hydropower flow channel, including all flow components such as turbine runner, guide-vanes, draft-tube, as well as upstream / downstream flow structures. He mentioned that the computational grid can be generated by a parametric grid-generator to have a fast and easy computational model build-up. Streamline flow through bulb turbine passage is shown in Fig. 9.

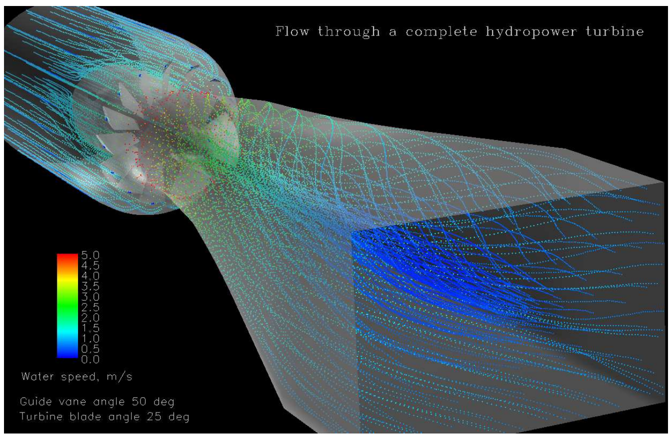

Fig. 9. Streamline flow through bulb turbine passage

V. CONCLUSION

Developments in CFD began with the use of much more simplified methods based on two dimensional potential theory, which did not take into account frictional effects. But nowadays CFD methods include three-dimensional, viscous and turbulence effects, rotor-stator interaction, multiphase flow, transient effects in the flow etc. The progress in CFD simulation technique is impressive and CFD is an obvious choice of designers of hydraulic turbines. Also, validity of CFD results is equally important. CFD can never replace the other two fundamental approaches of analysis i.e theoretical and experimental approach but it will supplement these two approaches and helps in reducing the experimental efforts and cost.

REFERENCES

[1] World Energy Report, International Energy Agency.
[2] M. Sick and A. Wilson "CFD for water turbines: review of the state of the art", Int J of Hydropower & Dams, Issue three, 2005, pp. 52-54.
[3] A. Ruprecht, " Unsteady flow analysis in hydraulic turbomachinery".
[4] B. Michel, M. Couston, M. Francois, and M. Sabourin, "Hydro turbines rehabilitation," IMechE Event Publications, v 2004 6, Hydropower Developments - New Projects, Rehabilitation, and Power Recovery– IMechE Conference Transactions, pp. 3-12.
[5] A Team from Suzler Hydro and Suzler Innotec, The article on "Design by numbers [hydraulic turbines]," International Water Power & Dam Construction, v 50, n 3, March 1998.
[6] S. Huang, Ph.D. dissertation on " Analysis and design of a new up-draft free-exit-flow low-head hydropower turbine system," 2000.
[7] G. D. Ciocan, M. S. Iliescu, T. C. Vu, B. Nennemann, and F. Avellan, "Experimental study and numerical simulation of the FLINDT draft tube rotating vortex", ASME J. Fluids Eng., 129, pp. 146-158, 2007.
[8] S. Jain, Dissertation on "CFD based flow analysis of hilly small hydropower station".
[9] Wang, Z., and Zhou, L., 2006, "Simulations and Measurements of Pressure Oscillations Caused by Vortex Ropes,", ASME J. Fluids Eng., 128, pp. 649-655.
[10] Keck, H., Drtina, P., and Sick, M., 1997, "Breakthrough - CFD flow simulation for a complete turbine," Sulzer Technical Review, v 79, n 1, pp. 26-29.
[11] Resiga, R. S., Muntean, S., Hasmatuchi, V., and Bernad, S., "Development of a swirling flow control technique for Francis turbines at partial discharge," 3rd German-Romanian workshop on Turbomachinery Hydrodynamics, May 10-12, 2007, pp.27-40.
[12]] J. Wu, K. Shimmei, K. Tani, K. Niikura, and J. Sato, "CFD-based design optimization for hydro turbines," ASME J. Fluids Eng., 129, pp. 159-168, 2007.
[13] Sick, M., Drtina, P., and Casey, M.V., 1998, "The use of stage capability in CFD for turbomachinery, with application in a Francis turbine," Int. J. Computer Applications in Technology, v 11, n 3-5, pp. 219-29.
[14] Sebestyen, A., Sallaberger, M., and Keck, H., 1997, "CFD-driven rehabilitation of hydraulic power plants," Proceedings of the Int. Conference on Hydropower - Waterpower, v 3, pp. 1719-1728.
[15] Bajic, B., 2002, "Multidimensional Diagnostics of Turbine Cavitation,", ASME J. Fluids Eng., 124, pp. 943-950.
[16] Cooper P., Mccormick M. & Worthen R., "Feasibility of using large Vertical Pumps as Turbines for small scale hydro power", Acc. 02, IDO-10109, prepared by EG & G Idaho Inc., September 1982, pp. B-3-1 to B-3-30.
[17] Tamm, A., "Analysis of a Standard Pump in Reverse Operation Using CFD".
[18] A. Shukla, R.P.Saini. "CFD Approach for PAT- A Review"
[19] US Army Corps of Engineers, Report on "Stay vane and wicket gate relationship study", Hydro Electric Design Center, 2005.
[20] Eisinger, R., and Ruprecht, A., "Automatic Shape Optimization Of Hydro Turbine Components Based On CFD".
[21] F. Avellan, "Introduction to cavitation in hydraulic machinery," 6th Int. Conference on Hydraulic Machinery and Hydrodynamics, Romania, pp. 11-22, Oct. 2004.
[22] G. E. Hecker, and T. C. Cook, "Development and evaluation of a new helical fish-friendly hydroturbine", ASCE J. Hyd. Eng., pp. 835-844, 2005".
[24] H. Pan, presentation on "Performance Prediction of Bulb-turbines by flow simulations".
[25] S. Jain, R.P.Saini, A.Kumar, " Review of flow analysis of hydro turbines using Computational Fluid dynamics", NUCONE 2007.