Review of Flow Analysis of Hydro Turbines Using Computational Fluid Dynamics

Sanjay Jain[1], R.P. Saini[2] and Arun Kumar[3]

Alternate Hydro Energy Centre, Indian Institute of Technology Roorkee

E-mail: [1]svja5pah@iitr.ernet.in, [2]rajsafah@iitr.ernet.in, [3]akumafah@iitr.ernet.in

***Abstract*—Flow analysis in hydro power generation has long been an intensive subject of research. For cost-effective design of different components, highly complex flow, *which is turbulent and three dimensional in nature*, is to be analyzed critically. The development of the high speed computer and evolution of the Computational Fluid Dynamics (CFD) have a great influence on the design and analysis of these components. This review examines the different investigations being carried out by different investigators in the field of hydro turbines using CFD as a numerical simulation tool. The specific problems related to different turbines are discussed. The latest trends for the costeffective design of hydro turbines are also discussed.**

***Index Terms*—Computational Fluid Dynamics, Flow Analysis, Hydro turbines, Part load operating conditions.**

I. INTRODUCTION

There is an increasing demand of energy in the world, with most of it from the developing countries. World energy demand is expected to nearly double by 2025. As the non-renewable fossil energy sources continues to deplete, and realizing the summits held at Brazil and Kyoto, to reduce the greenhouse gas emissions, hydro power has moved towards the top power development option to meet the increasing energy demand.

Flow analysis in hydro power generation has long been an intensive subject of research. As such, there are various components which comes in contact with the water, but turbine is the most critical component because it has a considerable influence on the cost of civil works as well as overall performance of any hydro power project. Turbines are usually designed according to the particular site conditions like head, discharge, silt content in the water etc. Hence, to achieve better performance with higher efficiency at lower cost, it is necessary to design the turbine according to the flow behavior in the turbine passage.

The flow in the hydro turbine is extremely complex due to its turbulent and three dimensional nature and rapidly changing curvature of the passage in the runner. In addition, it also exhibits unsteady behavior as a result of the interaction between rotating and stationary parts, i.e. runner and guide vanes. Considering these complexities, most of the investigators have analyzed the flow in hydro turbine using numerical analysis tool Computational fluid Dynamics (CFD), which reduces the time required for the design phase by predicting performance, efficiency, cavitation and hydrodynamic behavior accurately.

The continuous advancement in computational methods has offered hydro turbine designers a state-of-the-art technology for the design of turbine components. In the last two decades, the CFD technology has become an integrated part of the engineering design of hydro turbines. The attractiveness of the CFD technology is attributed to the following advantages associated with this advanced numerical methodology [1].

1. The turbo machinery flows can be more accurately predicted. This allows designers to explore more design alternatives, which would otherwise be too timeconsuming or are outside the range of previous experience.
2. The application of CFD in hydro design enables designers to have better control on the flow behavior. Higher turbine loading and flow capacity can be achieved without causing detrimental flow phenomena such as flow separation, cavitations and choking. In consequence, the size and therefore the cost of the turbine can be reduced.
3. The modern computer visualization of flow fields helps the turbine designer to understand the flow in turbine passages in better manner.
4. The numerical grids can reduce the turbine development time and costs. In some circumstances, the numerical simulation is satisfactory and can replace the model experiment, which is costly and time consuming.

For designers, prediction of operating characteristic curve is most important. All theoretical methods for prediction of performance merely give a value; but one is unable to determine the root cause for the poor performance. Using CFD analysis, one can get the performance value as well as observe actual behavior. Further, the root cause for the poor performance can also be find out.

It is the purpose of this review to examine the different investigations being carried out by different investigators in the field of Reaction and Impulse type hydro turbines using CFD as a numerical simulation tool. The specific problems related to different turbines are discussed. The latest trends for the cost-effective design of hydro turbines are also discussed.

II. Discussion of Considered Work for Reaction Turbines

The most commonly used reaction turbines, Francis turbine, *used for medium range of head and discharge*, and Kaplan turbine, *used for low head and high discharge*, being discussed. As mentioned, the turbines are designed according to site conditions, hence for the cost-effective design of turbines the flow through the turbine passage must be analyzed critically. Various researchers have carried out analysis for the cost effective design of turbine using CFD. Drtina and Sallaberger [2] discussed the basic principles of hydro turbines, with special emphasis on the use of CFD as a tool which is being increasingly applied to gain insight into the complex three-dimensional (3D) phenomena occurring in the turbines.

B. Michel *et al*. [3] described Alstom's experience in various hydro power stations of different countries, in the hydro turbine rehabilitation field. They found that, 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.

A team from Sulzer Hydro and Sulzer Innotec. [4] modelled a complete Francis turbine-*from the inlet of the spiral casing to the draft tube outlet*- using CFD which can be used to design new runners that match existing components more accurately, at a lower cost than by using model tests. Fig. 1 shows surface grids for entire Francis turbine and runner and guide vanes with the crown respectively.

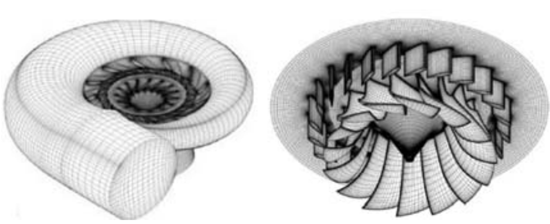

Fig. 1: Surface grids for Francis turbine (left) and runner and guide vanes (right)

Part Load Performance

The variable demand of the energy market, as well as limited energy storage capabilities, requires a great flexibility in operating hydro turbines. As a result, turbines tend to be operated far from the design flow conditions i.e. at part load conditions.

In Francis turbine, at part load operating conditions, turbine fixed-pitch runner shows a strong swirl at the runner outlet. As the incoming swirling flow decelerates in the draft tube, a hydrodynamic instability arises which leads to formation of strong helical vortex, rotating like a whirling rope, in the centre of the draft tube. The precession of the vortex rope causes pressure fluctuation in the draft tube which can lead to variation in power output, vibration of the shaft and damage to the runner blade. All these factors ultimately results into decrease in efficiency at part load operations, that’s why Francis turbine is having narrow operating range compared to Kaplan turbine, in which part load performance is comparatively better due to adjustable runner blades.

Many investigators have analyzed the part load behavior of Francis turbine. Yang Guo *et al*. [5] simulated the flow in the draft tube of a Francis turbine using CFD. They found that, under the part load conditions, the whirl of a vortex is produced in the draft tube and under certain operation conditions it may cause flow instability that leads to vibration or noise. They also mentioned that the operation range of the turbine is strongly related to the cavitation phenomena, which may occur either in the vanes of the runner or in the stationary parts.

Gabriel Dan Ciocan *et al*. [6] 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 simulated the flow and validated the same with experimental results. They carried out experimental investigations for a range of Thoma cavitation numbers varying from $\sigma = 1.18$, *cavitation free conditions*, to $\sigma = 0.38$, *maximum rope volume*. The vortex rope formation at different cavitation numbers is shown in Fig. 2.

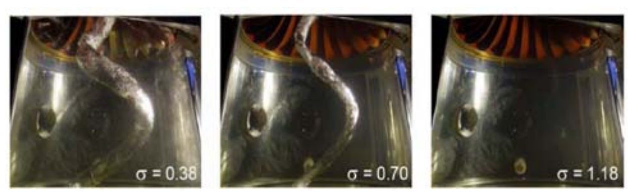

Fig. 2: Vortex rope formation at different cavitation numbers

Romeo Susan-Resiga *et al*. [7] carried out an experimental and theoretical investigation of the flow at the outlet of a Francis turbine runner in order to elucidate the causes of a sudden drop in the draft tube pressure recovery coefficient at a discharge near the best efficiency operating point. They analyzed the flow using CFD for a steady, axi-symmetric, and inviscid swirling flow and reveals that the swirl reaches a critical state precisely at the discharge where the sudden variation in draft tube pressure recovery is observed. They felt that for turbine design and optimization, suitable runner geometry should avoid such critical swirl configuration within the normal operating range.

Albert Ruprecht [8] carried out unsteady flow analysis of turbine using CFD. He discussed self excited unsteadiness, e. g. vortex shedding or vortex rope in the draft tube, as well as externally forced unsteadiness by changing or moving geometries, e.g. rotor-stator interactions. He accessed the requirements, potential and limitations of unsteady flow and discussed the demands of the turbulence models and the necessary computational efforts. The instantaneous flow for a certain time step is shown in Fig. 3, where an iso-pressure surface as well as the secondary velocity vectors in three cross-sections are plotted.

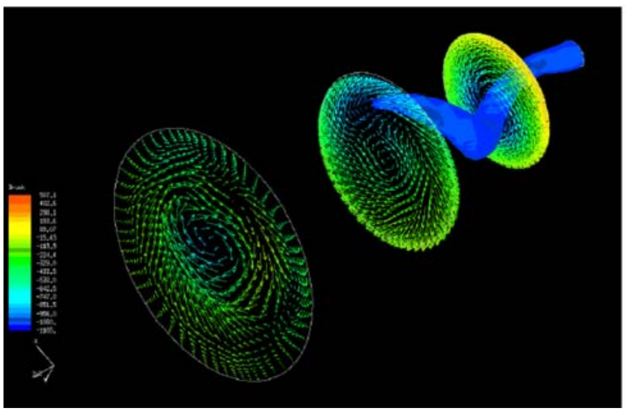

Fig. 3: Iso-pressure and secondary flow of a vortex rope

Some researchers have suggested different techniques to improve the overall performance of the turbine. Patel and Satanee [9] introduced the vanes in the draft tube of a Francis turbine and found that secondary flow, i.e. recirculation in the draft tube minimizes by providing vanes in the draft tube. They also found that by introducing the vanes in the draft tube, the head drop increases marginally in the casing, guide vanes and runner but it reduces drastically in the draft tube, as shown in Table 1.

Table 1: Summary of % Head Drop

Sr. No.	*Turbine component*	*Draft tube without vanes*	*Draft tube with vanes*
1.	Spiral casing and stay vane	1.03	1.04
2.	Guide vane	2.46	2.48
3.	Runner	94.46	95.15
4.	Draft tube	2.05	1.33

Romeo Susan-Resiga *et al*. [10] introduced a novel, simple and robust, method to mitigate the vortex rope by using a water jet issued from the tip of the crown cone in Francis turbine, as shown in Fig. 4. They supplied the jet with high pressure water from spiral case inlet, through the tubular shaft. They found elimination of severe pressure fluctuations at partial discharge, combined with a significant increase in the draft tube efficiency, which compensates the some percents of the overall turbine discharge that bypass the runner. Fig. 5(a) shows, *when the jet is not introduced*, a nicely developed single helical draft tube vortex rope and Fig. 5(b) shows, *when the jet is in operation*, the central low pressure region indicated by the iso-surface which is being greatly reduced and the shape of the vortex rope is changed from clearly helical to a slightly off-centre extended cone.

They presented the following advantages of this technique.

1. It successfully addresses directly the main cause of the flow instability, rather than the effects.
2. It does not require geometrical modifications of the runner, and no other devices needs to be installed in the draft tube.
3. It is continuously adjustable according to the operating point, and it can be switched-off when it is not needed.

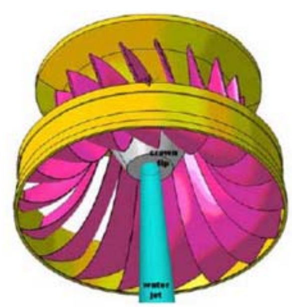

Fig. 4: Water jet injected at the crown tip.

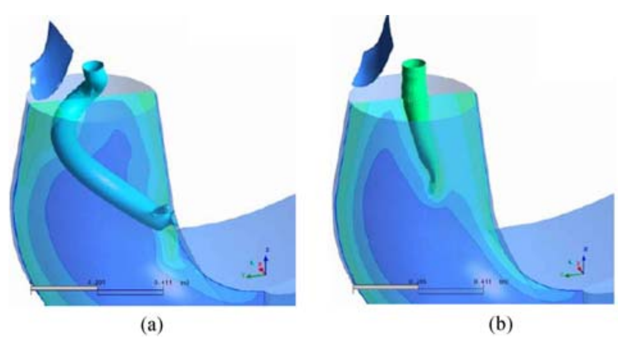

Fig. 5: Draft tube rope at part load conditions (a) without jet (b) with jet.

4. The practical implementation is simple and robust.
5. Although a fraction of the discharge bypasses the bladed region, the overall turbine efficiency does not suffer due to the improvement in both runner and draft tube efficiencies when the jet is on.

They evaluated the effectiveness of this technique quantitatively using CFD and found reduction in wall pressure fluctuations at part load, while the overall turbine efficiency remains practically unchanged.

Cavitation Analysis

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. Francois Avellan [11] 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.

Branko Bajic [12] 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 mentioned that:

1. How a turbine cavitation behavior can be improved? and
2. How a turbine operation can be optimized with respect to cavitation erosion?

Interaction between Different Components

In hydropower generation the push towards lower cost and more flexible output leads to the demand for more compact machines and a larger required operating range for new and upgraded turbines. As a result hydrodynamic effects in turbines such as the draft tube rope at part load and the guide vane-runner blade interaction become important issue in the design of these machines. Particularly in medium to high head Francis turbines, the pressure fluctuations become significantly large with respect to stress levels because in these machines the velocity at the guide vane outlet is sufficiently high and the radial gap between the blade rows tends to be small. 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. More recently the hydrodynamic guide vane-runner blade interaction has been found to be directly responsible for even blade cracking also. 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. B. Nennemann *et al*. [13] presented a method to simulate the unsteady flow field resulting from wicket gate-runner interaction in Francis turbine using CFD. They also validated the method by means of unsteady pressure measurements on the model of the runner. Fig. 6 shows rotor-stator interaction between blades and wicket gates of a Francis turbine and corresponding variations in flow and blade torque.

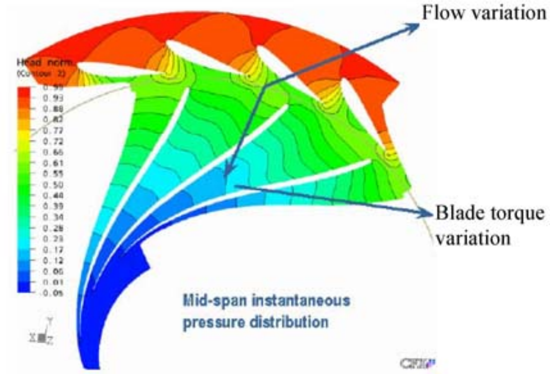

Fig. 6: Rotor-stator interaction between blades and wicket gates of a Francis turbine

Jingchun Wu *et al.* [14] conducted extensive turbulent flow simulations using CFD for both the existing and new designs of Francis turbine, at the 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. Fig. 7 shows comparison between measured efficiency-discharge (ç–Q) curves and predicted efficiency at target operation point for the new runner with existing tandem cascade, and the entire new optimized design as well.

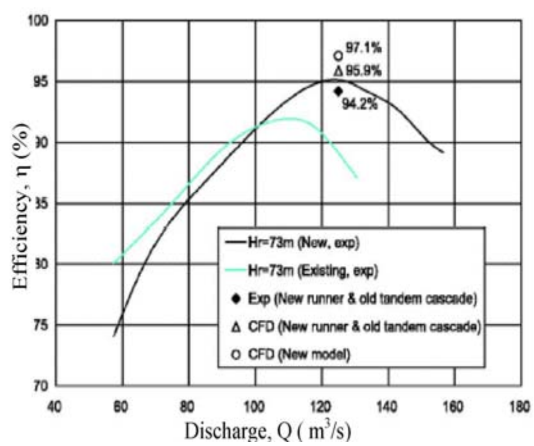

Fig. 7: Efficiency-discharge (ç–Q) curve

The team of US Army Corps of Engineers [15] 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.

Turbine Runner Improvement

The basic design of turbine runner should be based on the governing parameters from classical equations followed by the modifications using CFD analysis to improve the overall performance of the turbine.

Some researchers have carried out CFD analysis to improve the existing runner of the turbine. Hermod Brekke [16] introduced a runner with splitter blades for Francis turbine having longer blades and found that this runner reduces the danger of reversed flow on the pressure side of the blades near the inlet, *at part load*, due to the reduced distance between the blades at the inlet. He also described a philosophy behind pressure-balanced blades with skewed outlets which are made to stabilize dynamic behavior and to improve the efficiency during off design conditions.

S. Etter *et al*. [17] mentioned the benefits of chamfering the trailing edges of Francis turbine blades. They found that chamfering of the trailing edge of the runner blade and rounding of the blade-to-band joint can reduce the cavitation damage to the runner trailing edge and band, as well as it also gives more control over the frequency and intensity of the vortex wake. They also mentioned that thinner and less intense vortex street issuing from the chamfered trailing edge has potential for improvement in turbine efficiency in the 70 to 90 percent output range.

Fish Friendly Turbines

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*. [18] 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%.

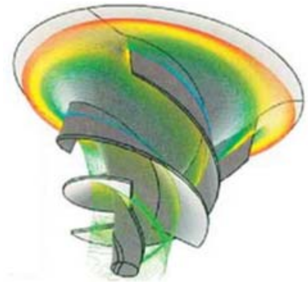

Fig. 8: Fish-friendly Kaplan turbine

Silt Erosion

The sediment content in rivers may cause abrasion in different components of hydro turbines. The erosion intensity depends on the sediment type, its characteristics (particle size distribution, shape and quantity), on the operating condition of the machine (flow rate, head and rotation speed), the hydro design itself, as well as the material used for the turbine components. It may also leads to decrease in the efficiency of the turbine. Stephen Bergeron *et al*. [19] simulated the flow in the Francis turbine using CFD and identified the locations of erosion risk on the different components as well as the erosion intensity from which the erosion depth can be predicted.

Boundary Layer Development

In Francis turbine the evolution of the boundary layer in the cone is complex due to the rotating flow at the runner outlet, the adverse pressure gradient, the interaction with the leakage flow and the unsteady perturbations due to the vortex rope. E.L. Berca *et al*. [20] developed the experimental setup for wall friction measurements and near wall velocity measurements in the cone of a Francis turbine scale model. They characterized the boundary layer for an operating range in the vicinity of the best efficiency point. They compared the results with the Karman-Prandtl model and also mentioned the reasons for the variations in the results.

III. Discussion of Considered Work for Impulse Turbines

The most commonly used Impulse turbine, Pelton turbine, *used for high head and low discharge*, being discussed. Performance prediction of hydro turbines, such as efficiency and dynamic behavior under different operating conditions, is of high interest to manufacturers. Up to now, Pelton turbines have been designed using experimental techniques and semiempirical methods. The reason is that the flow in the bucket is unsteady, separated from air by an unknown free surface (two-phase flow), and developed within moving boundaries. In today's highly competitive market of turbine upgrading and refurbishment, the performance guarantees are often difficult to determine in the short term. An accurate prediction of performance of Pelton turbine by CFD would reduce the time required for the design phase. Mnay investigators have carried out CFD based analysis of Pelton turbine.

Silt Erosion

T.R. Bajracharya *et al*. [21] studied the effect of silt erosion on the Pelton turbine bucket and nozzle. They performed erosion analysis, sediment load calculations and drawn the flow net diagram for the flow analysis through the surface of needle and found that, the estimated wear rate of

3.4 mm/yr for the needle and the bucket led to efficiency reduction of 1.21%. Fig. 9 shows erosion of bucket and needle respectively.

Fig. 9: Erosion of bucket (left) and needle (right)

Flow Analysis

Alexandre Perrig *et al*. [22] carried out flow analysis for the backside of the Pelton turbine bucket using CFD and found that the bucket backside contributes to the bucket torque which can be linked with the interaction between the water jet and the bucket cutout profile. The jet appears to adhere to some extent to the bucket back, showing the presence of a Coanda effect which leads to the creation of a lift force, and from that an angular momentum in the sense of rotation, contributing to the runner torque.

B. Zoppe *et al*. [23] carried out detailed experimental and CFD analysis of the flow in a fixed bucket of a Pelton turbine by taking different head, jet incidence, and flow rate, to cover a wide range of the turbine functioning points. They found that, the jet trajectory inside the bucket do not influenced by variation in head and a leakage flow through the cutout rapidly increases with the jet diameter and the bucket incidence. Fig. 10 shows computational mesh for bucket and free surface of the jet in the bucket respectively.

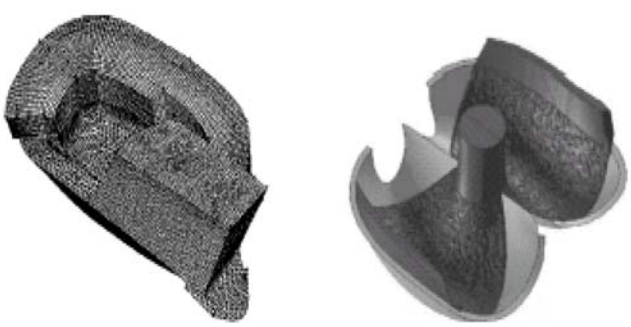

Fig. 10: Computational mesh for bucket (left),
free surface of the jet in the bucket (right)

Filip Sadlo *et al.* [24] presented the different roles of vorticity by analyzing the visualization of vortices, its distribution and role in vortex phenomena, using CFD. They also mentioned that quality of the water jets in Pelton turbine is affected mostly by vortices originating in the distributor ring. Hence, for a better understanding of this interrelation, it is crucial to not just visualize these vortices but also to analyze the mechanisms of their creation.

IV. CONCLUSIONS

CFD has been used to analyze various fluid flow related problems in various sectors. In hydro power sector, CFD is being used by different investigators to analyze flow behavior in different turbines. It is found that in Reaction turbines CFD is used to analyze the flow for the following:

- To analyze the flow at part load operating conditions.
- To analyze cavitation behavior.
- To study the interaction between different components.
- To improve the runner profile.
- To develop fish friendly turbines.
- To study the effect of silt erosion.
- To study the effect of boundary layer development.

In case of Impulse turbines, CFD has been used for the following:

- To analyze the flow on the backside of the bucket.
- To study the jet trajectory inside the bucket.
- To study the effect of vorticity on the quality of the water jet.
- To study the effect of silt erosion on the bucket and the needle.

It is found that, in the hydro turbine rehabilitation field, runner and guide vanes can be replaced easily but modifications affecting the mechanical structure of the turbine such as stay vanes or the concreted parts such as spiral case or draft tube, is difficult. However, to achieve the better performance, extensive CFD based analysis followed by model test, is generally necessary.

REFERENCES

[1] Huang, S., Ph.D. dissertation on "Analysis and design of a new up-draft free-exit-flow low-head hydropower turbine system," 2000.

[2] Drtina, P., and Sallaberger, M., "Hydraulic turbines-basic principles and state-of-the-art computational fluid dynamics applications," Proceedings of the Institution of Mechanical Engineers, Part C (Journal of Mechanical Engineering Science), v 213, n C1, pp. 85-102, 1999.

[3] Michel, B., Couston, M., Francois, M. and Sabourin, M., "Hydro turbines rehabilitation," IMechE Event Publications, v 2004 6, Hydropower Developments—New Projects, Rehabilitation, and Power Recovery– IMechE Conference Transactions, pp. 3-12.

[4] 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.

[5] Guo, Y., Kato, C., and Miyagawa, K., "Large-eddy simulation of non- cavitating and cavitating flows in the draft tube of a Francis turbine".

[6] Ciocan, G.D., Iliescu, M.S., Vu, T.C., Nennemann, B. and Avellan, F., "Experimental study and numerical simulation of the FLINDT draft tube rotating vortex", ASME J. Fluids Eng., 129, pp. 146-158, 2007.

[7] Resiga, R.S., Ciocan, G.D., Anton, I. and Avellan, F., "Analysis of the swirling flow downstream a Francis turbine runner," ASME J. Fluids Eng., 128, pp. 177-189, 2006.

[8] Ruprecht, A., "Unsteady flow analysis in hydraulic turbomachinery". [9] K. Patel, and M. Satanee, "New development of high head Francis Turbine at Jyoti ltd. for small hydro power plant," Proceedings of Himalayan Small Hydropower Summit (HSHS), Dehradun, 2006.

[10] Resiga, R.S., Vu, T.C., Muntean, S., Ciocan, G.D. and Nennemann, B., “Jet control of the draft tube vortex in Francis turbines at partial discharge,” 23rd IAHR Symposium, Yokohama, Oct. 2006.

[11] Avellan, F., “Introduction to cavitation in hydraulic machinery,” 6th Int. Conference on Hydraulic Machinery and Hydrodynamics, Romania, pp. 11-22, Oct. 2004.

[12] Bajic, B., “Multidimensional diagnostics of turbine cavitation,” ASME J. Fluids Eng., 124, pp. 943-950, 2002.

[13] Nennemann, B., Vu, T.C. and Farhat, M., “CFD prediction of unsteady wicket gate-runner interaction in Francis turbines: A new standard hydraulic design procedure”.

[14] Wu, J., Shimmei, K., Tani, K., Niikura, K. and Sato, J., “CFD-based design optimization for hydro turbines,” ASME J. Fluids Eng., 129, pp. 159-168, 2007.

[15] US Army Corps of Engineers, Report on “Stay vane and wicket gate relationship study”, Hydro Electric Design Center, 2005.

[16] Brekke, H., “State of the art in turbine design”.

[17] Etter, S., Otto, A. and Gummer, J.H., “Benefits of chamfering the trailing edges of Francis turbine blades,” Hydropower and Dams, Issue 2, pp. 68- 72, 2007.

[18] Hecker, G.E. and Cook, T.C., “Development and evaluation of a new helical fish-friendly hydroturbine”, ASCE J. Hyd. Eng., pp. 835-844, 2005.

[19] Bergeron, S., Vincent, A. and Vu, T.C., “Silt erosion in hydraulic turbines: the need of real time numerical simulation,” The Society for Computer Simulation International, 1998.

[20] Berca, E.L., Ciocan, G.D. and Avellan, F., “Wall friction and boundary layer development in the cone of a Francis turbine scale model,” 22nd IAHR Symposium on Hydraulic Machinery and Systems, Sweden, June 2004.

[21] Bajracharya, T.R., Acharya, B., Joshi, C.B., Saini, R.P. and Dahlhaug, O.G., “Sand erosion of Pelton turbine nozzles and buckets: A case study of Chilime hydropower plant,” ELSEVIER (Article in Press), 2007.

[22] Perrig, A., Avellan, F., Kueny, J.L., Farhat, M. and Parkinson, E., “Flow in a Pelton turbine bucket: numerical and experimental investigations,” ASME J. Fluids Eng., 128, pp. 350-358, 2006.

[23] Zoppe, B., Pellone, C., Maitre, T. and Leroy, P., “Flow analysis inside a Pelton turbine bucket,” ASME J. Turbomachinery, 128, pp. 500-511, 2006.

[24] Sadlo, F., Peikert, R. and Parkinson, E., “Vorticity based flow analysis and visualization for Pelton turbine design optimization,” appeared in the Proceedings of IEEE Visualization.