Research Journal of Engineering Sciences ISSN 2278 – 9472
Vol. **6(5),** 7-12, June **(2017)** *Res. J. Engineering Sci.*

Study on flow characteristics of radial distributor of hydraulic turbine

Ruchi Khare[1*] and Mithun Gole[2]
[1]Dept. of Civil Engineering, MANIT, Bhopal, MP, India
[2]Hydro-Power Engineering, MANIT, Bhopal, MP, India
ruchif4@rediffmail.com

Available online at: www.isca.in, www.isca.me
Received 8th April 2017, revised 5th June 2017, accepted 18th June 2017

Abstract

The frequent change in electrical load on generator leads variation of speed of generator during operation of hydro power plant and thus affects the frequency. In hydraulic turbine, distributor is used to regulate and direct the water to runner. Hence it is required to maintain constant speed of generator to have constant frequency output. As hydraulic turbines are directly coupled to generator and to maintain constant speed, the discharge through turbine is varied. The radial distributors are axi-symmetric vanes which control and feed water to runner at desired angle. In present work, numerical flow simulation of radial distributor of a mixed flow Francis turbine is done for different shape of guide vane (distributor) and their opening. The flow parameters are computed by using results obtained from simulation and the effect of shape of guide vanes at different guide vane openings on flow parameter is analysed.

Keywords: Guide vane, Radial distributor, Efficiency, Performance characteristic.

Introduction

In mixed flow Francis turbine, water passes from reservoir to penstock, spiral casing, stay ring, distributor, runner and draft tube. The flow through stationary casing, stay ring, distributor, rotating blade rows and diffuser becomes so complex that theoretical analysis is not possible. The turbines are designed for fixed values of global and local design parameters but most of the time turbine operates at part load conditions which affect its performance.

For economical turbine design, it is very necessary to analyse the flow characteristics in different parts of the turbine, which may help in predicting their performance before manufacturing. The overall performance of the turbine depends on the performance of its different components. The change in profile of radial distributor leads to variation in torque, flow angle and flow direction, which in turns leads to the variation of flow parameters in different components of the turbine.

The radial distributor consists of a series of axi-symmetric vanes which control and feed water to runner at desired angle. The entry of water to runner is smooth only for a fixed guide vane opening and the flow of water to runner other than this angle will cause impact losses. The pressure and velocity distribution on the guide vanes of distributor depends on the shape of guide vane profile and their opening. For the efficient design of distributor the pressure and velocity distribution are required at different guide vane opening.

With the advent of high speed digital computer and super computer along with the development of accurate algorithm enabled us to solve large scale physical problem based on numerical solution. This has introduced fundamentally an important approach - "The approach of Computational Fluid Dynamics (CFD)" i.e. the methodology to find out solutions of problems with numerical techniques (FVM, FDM, FEM). The computational fluid dynamics is now an useful tool for flow simulation within the turbine component. However it never replaces the experimental approach but it serves as compliment to experimental approach.

Many researchers have already worked on the number of distributors and shape of distributor s of axial and mixed flow turbines[1-3]. Some of them regulated the discharge by appropriate change in distributor opening and analysed the local and global performance of turbine[4,5]. It is observed that the profile of distributor not only affects the flow distribution in the runner but also the overall performance of turbo machine[6].

In this paper numerical simulation of mixed flow Francis turbine is done by using five different NACA profiles of distributor. The local and global performance of turbine is compared in each case. It is found that high and low pressure sides of the distributor profile depends on positive or negative curvature of the profiles. The geometry of modelled turbine is prepared in CATIA P3 and ANSYS workbench software the numerical analysis is done by using ANSYS 16.0 software.

Geometric modeling

The geometry of the existing turbine is prepared in in ANSYS workbench and CATIA P3 as shown in Figure-1. It is a horizontal axis Francis turbine with number of stay vane 18, number of guide vanes 18, number runner of blades as 13 and draft tube of conical elbow shape.

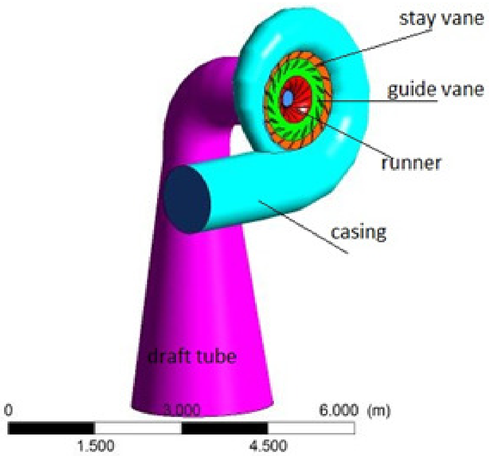
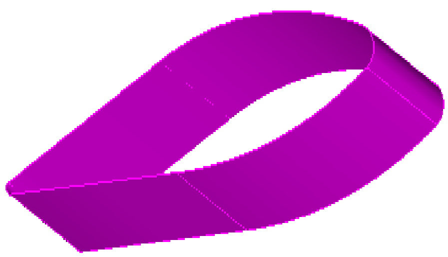


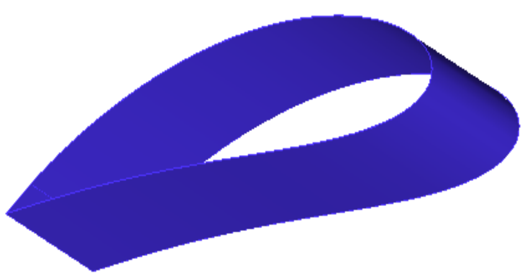
Figure-1: Complete Assembly of Francis Turbine.

The existing profile of distributor is symmetrical (Figure-2(a)). Apart from it four other positive and negative profiles as shown in Figure-2 (b, c, d, e) are also used for the numerical simulation.

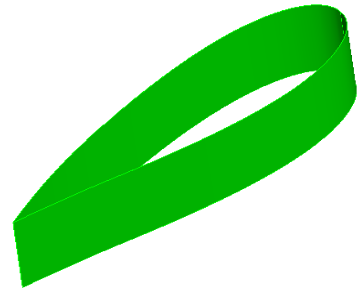


(a) Symmetric GV Profile.

(b) GV with NACA PC 6523 Profile.



(c) GV of NACA PC 7524 Profile



(d) GV of NACA PC 2722 profile.

(e) GV of negative camber NC 7524 profile.

Figure-2: Different types of distributor (GV) profiles.

The meshing of flow domain is done using combination of tetrahedral and prismatic elements. The meshing of the flow domain of the turbine was done by using ANSYS ICEM CFD (Figure-3). The boundary layer around the guide vane and runner blades was captured by prismatic layer and tetrahedral meshing was done for remaining flow domain. The grid dependency test at best operating point was carried out to minimize the influence of grid number on the computational results. The mesh size which gave nearest values of efficiency and output as obtained from the experimental results is selected. The detail of meshing in flow domain in case of each guide vane profile is given in Table-1.

Boundary conditions: The specified boundary conditions are mass flow rate of 7200 kg/s at inlet and static pressure at outlet. The reference pressure in each case is considered as 1 atm. The steady flow simulation is done by using SST k-ω turbulence model has been used for simulation at high resolution scheme. The walls of the turbine are kept as smooth with no slip condition[7,8]. The analysis is carried out for constant discharge and three different rotational speeds i.e, 500 rpm, 600 rpm and 700 rpm. The best operating condition of the existing turbine is at mass flow rate of 7200kg/sec at the rotational speed of 600 rpm.

Table-1: Mesh statics.

Types of GV profiles	No. of nodes	No. of elements
Symmetrical profile	1011273	5027514
NACA PC -6523 with positive camber	998775	4961102
NACA PC -7524 with positive camber	994329	4935839
NACA PC -2722 with positive camber	1005147	4999783
NACA NC 7524 wth negative camber	1012362	5039406

List of formulae: Formulae used for computing loss and flow parameters in non-dimensional form are:

Head (m) = $\frac{TPi-TPo}{\gamma}$ (1)

Power at Inlet = ϒQH (2)

Power at outlet = $\frac{2\pi NT}{60}$ (3)

Efficiency η = $\frac{outlet\ power}{inlet\ power}$ (4)

Total Head Loss $H_l = H_{lc} + H_{ls} + H_{lgv} + H_{lr} + H_{ldt}$ (5)

Head loss at distributor (GV) $\xi_{lgv} = \frac{H_{lgv}}{H} X100$ (6)

Head loss at runner $\xi_{lr} = \frac{H_{lr}}{H} X100$ (7)

Nomenclature: H - Net Head of turbine, H_{lc} - Head loss at casing of turbine, H_{ls} - Head loss at stayring of turbine, H_{lgv} - Head loss at distributor (GV) of turbine, H_{ld} - Head loss at draft tube of turbine, n - Speed of rotation of runner (rpm), TPi - Total pressure at casing inlet (pa), TPo - Total pressure at draft-tube outlet (pa), ϒ – Specific weight of water.

Results and discussion

The numerical analysis is done for flow regime of mixed flow Francis turbine by changing five different guide vane profiles. Each geometry was analysed at three rotational speeds and constant discharge. The variations in losses and efficiency are depicted through bar chart as shown in Figure-3 to Figure-5.

The variation of efficiency at different guide vane profiles and rotational speeds are shown in Figure-3. The best efficiency i.e, 85.92% is obtained by using PC-2722 profile of the guide vane at the speed of rotation as 600 rpm. The high efficiency is achieved at rotational speed of 600 rpm of runner in all distributor profiles except in case of NC-7594. It is because of adverse pressure variations in this guide vane (distributor) profile condition[9].

The head losses at distributor (guide vane) are shown in Figure-4. It is seen from figure that least head losses are observed in case of NC-7524 profile at all three rotational speeds. But efficiency is lower in this case because of generation of swirl at runner outlet, which creates more losses in rotating runner and the inlet portion of draft tube[10]. Figure-5 shows head losses at the runner at different profile of runner. The loss is more in case of NC-6424 profile but is least at PC-7524 profile of the guide vane that's why the higher efficiency is achieved in case of PC-7524 profile.

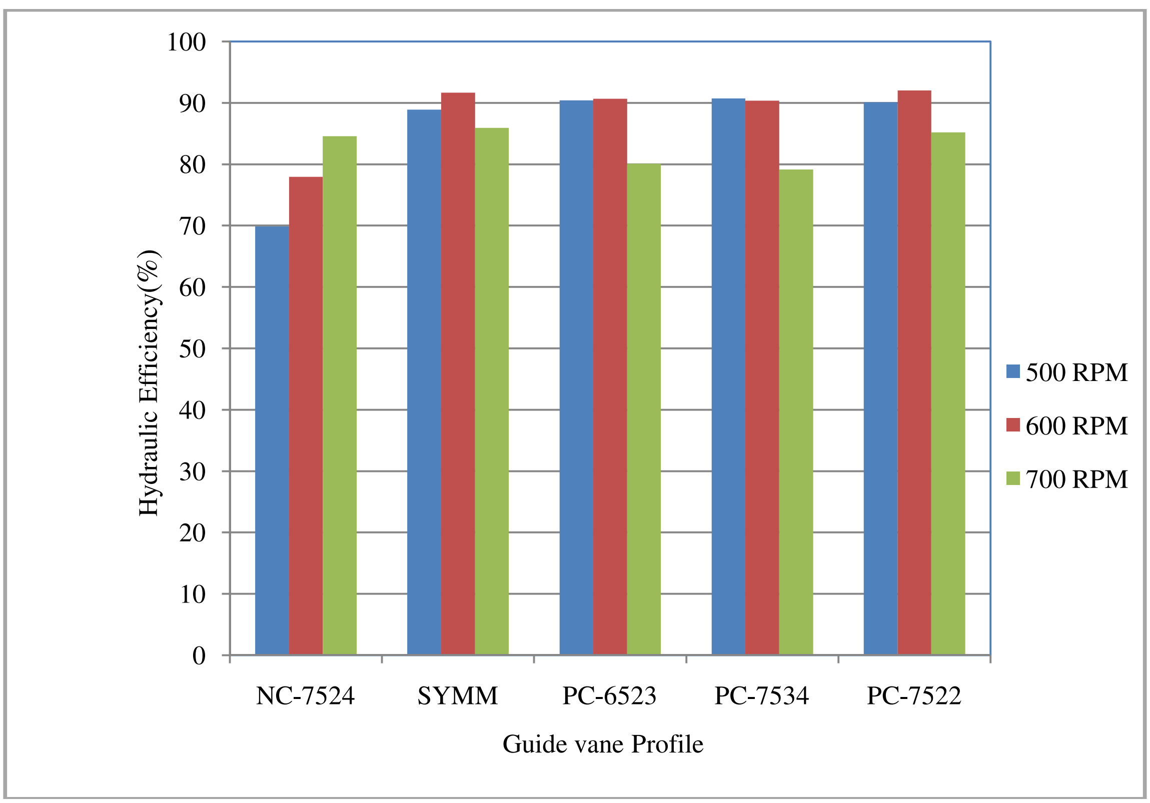


Figure-3: Overall efficiency variations at different guide vane profiles.

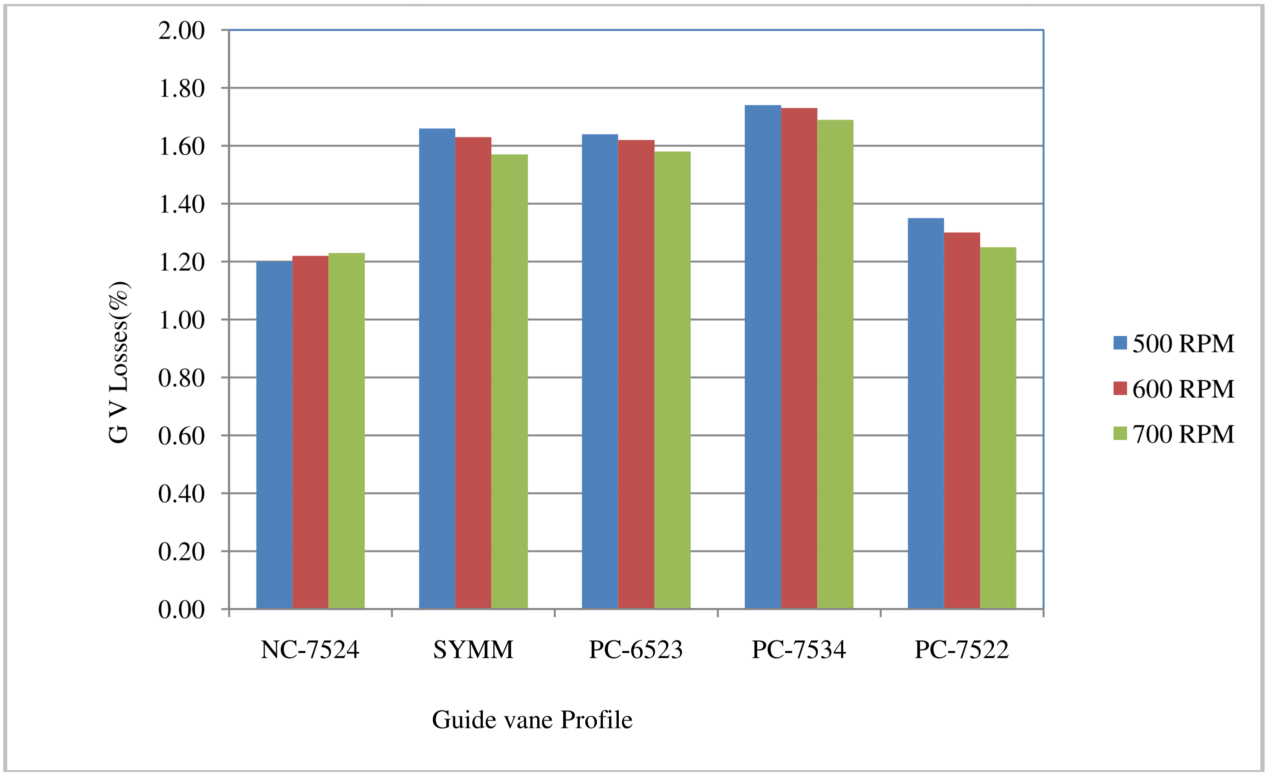


Figure-4: Head loss at distributor (Guide Vane).

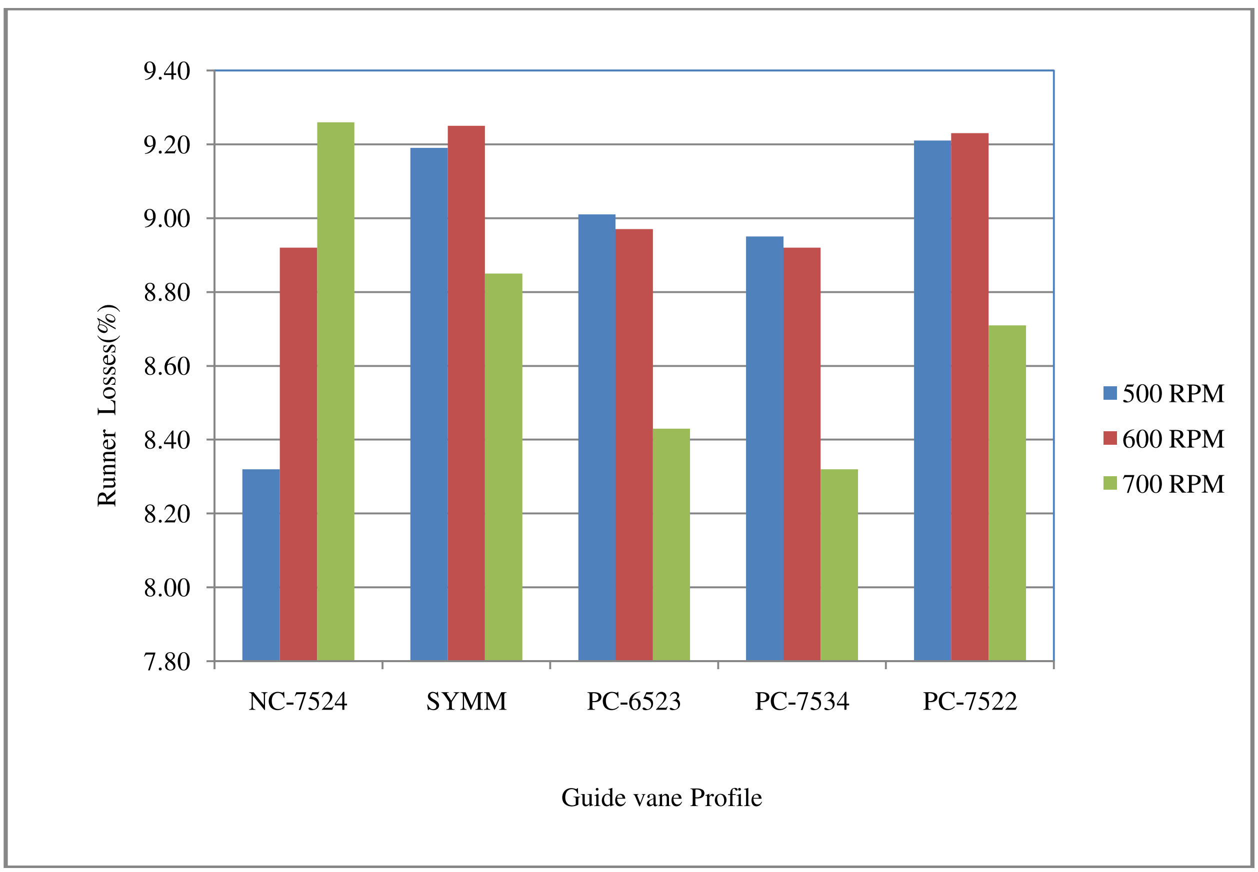
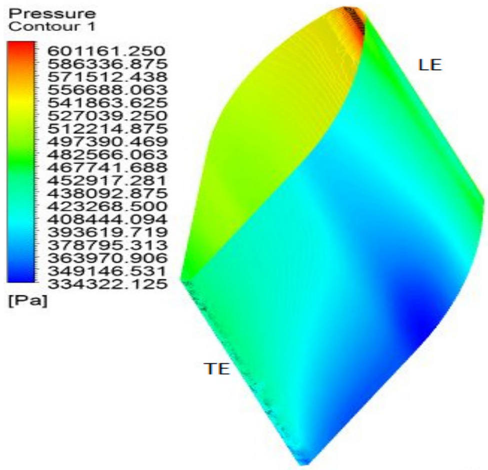
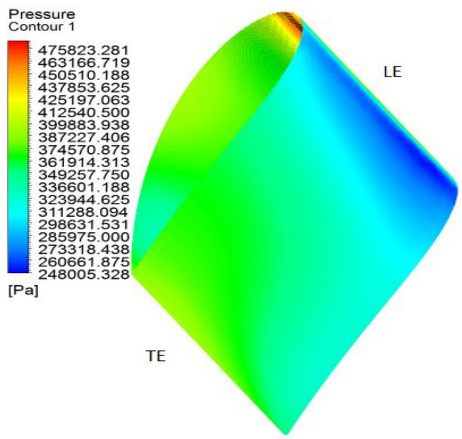


Figure-5: Head loss at runner.

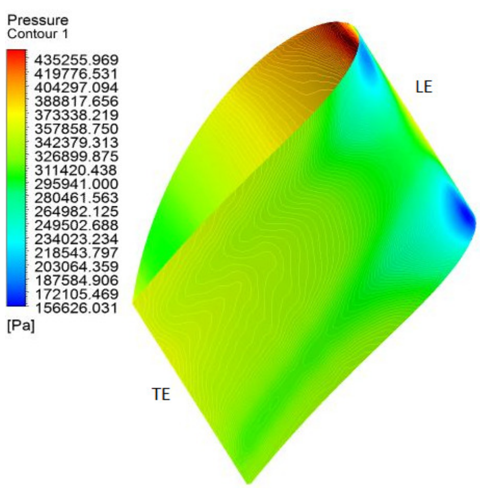
The variation of pressure from inlet to oulet in each blade of radial distributor is shown in Figure-6. The pressure at all profiles of guide vane is decreasing from inlet to outlet. It is because of conversion of pressure energy into kinetic energy[11].



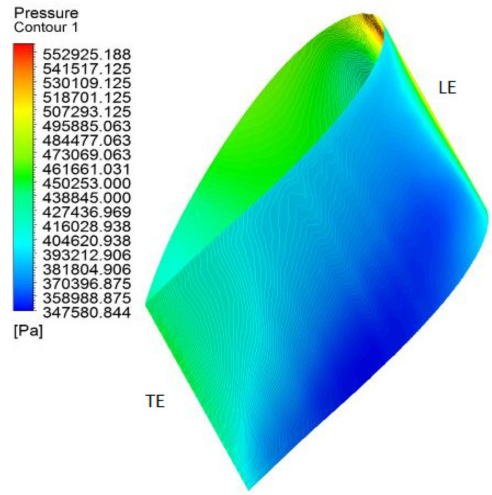
(a) Symmetric Profile



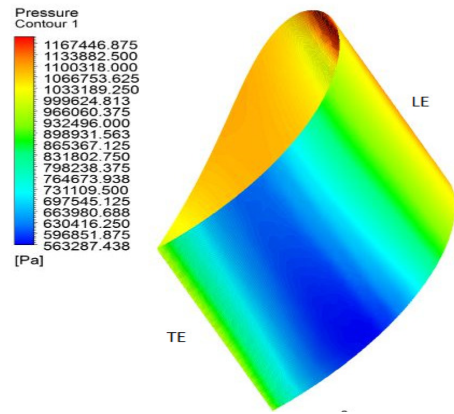
(b) GV with NACA PC 6523 Profile.



(c) GV of NACA PC 7524 Profile.



(d). GV of NACA PC 2722 profile.



(e) GV of negative camber NC 7524 profile.

Figure-6: Pressure contours at different distributor profiles at 600 RPM.

It can be observed that the range of pressure is varying with camber and profile of guide vane. The pressure contours show that the pressure variation is very large in case of NC-7524 profile.

The pressure is dectreasing due to acceleration of flow from inlet to outlet. The pressure variation is smooth in case of symetrical and PC-7524 profiles. As changing the camber of guide vane profile, pressure at leading edge varies, pressure at leading edge of guide vane depends upon camber and maximum camber position. With negative camber guide vane profile pressure at leading edge of guide vane goes to high as compare to positive guide vane profile.

Conclusion

The numerical flow simulation of mixed flow Francis turbine by using different shapes of profiles of guide vane reveals that the existing symmetrical profile of symmetrical guide vane can be replaced by PC-7524.

Changing the profile of distributor may be helpful in achieving higher efficiency from the same turbine. Numerical simulation of Francis turbine is cost effective and least time consuming solution for its design improvement.

References

1. Keck H. and Sick M. (2009). Thirty Years of Numerical Flow Simulation in Hydraulic Turbomachines. *Acta Mechanica*, 201(1-4), 211-229.
2. Daneshkah K. and Zangeneh M. (2010). Parametric Design of a Francis Turbine By Means of a Three Dimensional Inverse Design Method. 25th IAHR Symposium on Hydraulic Machinery and Systems, *IOP Conf. Series: Earth and Environmental Science,* 12, 1755-1315/12/1/012058.
3. Patel K., desai J., Chauhan V. and Charnia S. (2011). Development of Francis Turbine Using C.F.D. The 11th Asian International Conference on Fluid Machinery and The 3rd Fluid Power Technology Exhibition, IIT Madras, Chennai, India, Paper ID: AICFM_TM_015.
4. Khare R., Prasad V. and Kumar S. (2010). Effect of Operating Regime on Hydrodynamic Parameters of Mixed Flow Hydraulic Turbine. *Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, India*, FMFP10-TM-16.
5. Ayancik F., Aradag U., Ozkaya E, Unver O. and Aradag S. (2013). Hydroturbine Runner Design and Manufacturing. *International Journal of Materials, Mechanics and Manufacturing*, 1(2), 162-165.
6. Viscanti N., Pesatori E. and Turozzi G. (2009). Improvement of a Francis Runner Design. 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Brno, Czech Repulic, 177-184.
7. Rajoo S. and Martinez R. (2008). Mixed Flow Turbine Research: A Review. *ASME Journal of Turbomachinery*, 130(4), 044001-10.
8. Risberg S., Jonassen M. and Jonassen R. (2008). Design of Francis Turbine Runners Based on a Surrogate Model Approach. *International Journal of hydropower & Dams*, 5.
9. Muntean S., NINACI Ioan, SUSAN-RESIGA Romeo, BAYA Alexandru and ANTON Ioan (2010). Numerical Analysis of The Flow in the Old Francis Runner in Order to Define the Refurbishment Strategy. *U.P.B.Scientific Bulletin, Series D,* 72(1), 117-124.
10. Balint D., Campian V., Nedelcu D. and Megheles O. (2012). Hydrodynamics Automatic Optimization of Runner Blades for Reaction Hydraulic Turbines. *26th IAHR Symposium on Hydraulic Machinery and Systems, IOP conf. Series: Earth and Environment Science,* 15, 032014.
11. Enomoto Y., Kurosawa S. and Kawajiri H. (2012). Design Optimization of a High Specific Speed Francis Turbine Runner. *26th IAHR Symposium on Hydraulic Machinery and Systems, IOP conf. Series: Earth and Environment Science,* 15, 032010.