

 <p>ISSN NO. 2320-5407</p>	<p>Journal Homepage: - www.journalijar.com</p> <h2 style="text-align: center;">INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)</h2> <p style="text-align: center;">Article DOI: 10.21474/IJAR01/4209 DOI URL: http://dx.doi.org/10.21474/IJAR01/4209</p>	
---	--	---

RESEARCH ARTICLE

DESIGN AND PERMANENCE ANALYSIS OF FRANCIS TURBINE FOR HYDRO POWER STATION ON KUNAR RIVER USING CFD.

Wahidullah Hakim Safi¹ and Dr. Vishnu Prasad².

1. PG Scholar, Maulana Azad National Institute of Technology Bhopal, Bhopal – 462003 India.
2. Professor, Maulana Azad National Institute of Technology Bhopal, Bhopal – 462003 India.

Manuscript Info

Manuscript History

Received: 17 March 2017
Final Accepted: 15 April 2017
Published: May 2017

Key words:-

Computational Fluid Dynamics (CFD) analysis, Hydraulic turbine, Numerical simulation.

Abstract

Theoretical analysis of turbines for predicting of operating characteristics is a complex and poor way which gives only values and details of flow field behavior and causes for loss of efficiency cannot be investigated. Computational Fluid Dynamics (CFD) analysis is a robust technique for prediction of performance characteristics of hydraulic machineries and the actual fluid flow behavior can be observed. In this study the CFD analysis of a Francis hydraulic turbine is carried out which was selected for operation at a hydropower station on Kunar River in Afghanistan during some theoretical designs and calculations. Francis turbine is simulated based on designed data and operating behavior of turbine is predicted and fluid flow in turbine domain is observed.

Copy Right, IJAR, 2017. All rights reserved.

Introduction:-

Afghanistan is among the lowest in electricity usage about 100 kWh per capita per year. Before 2009, only 10-15% of population had access to electricity and now electricity is extended to 30% of the population. Peak supply in 2014 was 750 MW with an untapped estimated demand of 2500 MW which is increasing around 25% per year in major cities. About 80% electricity was supplied by imported power from neighboring countries. Afghanistan needs to become energy independent and to increase generation capacity within the country. The main sources of generation are hydropower, fossil fuel and renewable energy resources. (RRP AFG 47282-001[12]).

Hydro power in Afghanistan can be a key element for economic growth through management of available water resources of the country for energy production. It is found from analysis and planning of water resources of county that 800 MW hydro power station can be developed on Kunar River at Kunar province of Afghanistan. Kunar River is a main tributary of Kabul River basin which located in eastern Afghanistan and northern Khyber Pakhtunkhwa. Kunar River feeds from melting of glaciers and snows of Hindu Kush Mountains (Wahidullah H S et al [18]). Design of hydro power station on Kunar River included planning of reservoir, selection and design of hydro mechanical equipment, planning and layout of hydro power station. Arc-GIS is used to determine elevation volume and elevation surface area characteristics of reservoir (Wahidullah H S et al [18]). Dead reservoir storage and sedimentation of Kunar River is evaluated with HEC-RAS sediment model (Wahidullah H S et al [19]) and finally required storage and firm demand is obtained. Hydro power station is planned with a layout behind of concrete gravity dam taking all required auxiliaries of hydropower station.

Corresponding Author: - Wahidullah Hakim Safi.

Address: - PG Scholar, M. Tech, Hydropower Engineering, Department of Civil Engineering, Maulana Azad National Institute of Technology Bhopal, Bhopal – 462003 India.

Efficiency of a hydro power station depends on efficiency of individual components such as turbine, draft tube and generator. Enhancements in efficiency and hydro-dynamic design of a hydro power station causes higher supply of electrical power which represent a considerable economic value. (Dr. Vishnu Prasad, [1]). According to recent studies turbine efficiency can be improved with better understanding of complex flow field in the runner blades and introducing moderate modification to runner blades.

Three dimensional real flow analysis is done for an experimentally tested Francis turbine with horizontal rotation axis and operating characteristics of prototype were predicted in actual operating regime and compare with experimental results (Dr. Vishnu Prasad et al [2]). Numerical simulation of complete Francis turbine including unsteady stator-rotor interactions is performed with full generic model and no periodicity. The simulated entire model of Francis turbine and determined differences between single blade and complete turbine design (Ruprecht et al [4]).

CFD can be used for developing of new hydraulic turbines and controlling of alternate designs for turbine efficiency before experimental optimization and testing of selected designs. Performance analysis of designed hydraulic Francis turbine for mentioned hydro power station is carried out using ANSYS CFD code to predict flow and loss characteristic and fluid flow behavior.

Turbine design data:-

The five units of low specific speed Francis turbine was selected based on specific speed, head and discharge. The location and layout of hydro power station was fixed based on available discharge and head.

Steel scroll casing, stay vanes, guide vanes, runner, elbow type of draft tube of turbine were designed using standardized dimensions. Basic designed data and Francis turbine specifications are given in Table 1.

Table 1:- Design data for simulation of Francis turbine

No	Required parameters	Designed values
1	Type of turbine runner	Normal Francis Runner
2	Designed head (m)	289.17
3	Maximum head (m)	339.5
4	Minimum head (m)	171.3
5	Designed discharge (m^3/sec)	68.53
6	Designed Power of Turbine (MW)	112
7	Inlet diameter of turbine (m)	3.6
8	Discharge diameter of turbine (m)	2.7
9	Specific speed of turbine	140.358
11	Designed speed of turbine (rpm)	500
13	Number of runner blades	17
14	Diameter of guide vane to trailing edge (D_0) in (m)	4.3
15	Length of guide vanes (m)	1.0
16	Number of guide vanes	24
17	Length of stay vanes (m)	1.0
18	Number of stay vanes	12
19	Inlet flow angle (α_1) degree	28.42
20	Inlet runner blade angle (β_1) degree	15
21	Outlet flow angle (α_2) degree	90
22	Outlet runner blade angle (β_2) degree	13.29
23	Stay vane blade angle in degree	20
24	According to United States Bureau of Reclamation (USBR) manual for reaction turbines single passage elbow type draft tube is selected for turbine.	

Geometrical modeling of turbine:-

The geometrical modeling of stay vane, guide vane, runner and draft tube of Francis turbine is done with ANSYS Blade-Gen and ICEM CFD software for flow simulation. The geometries of turbine components are shown in figure 1.

Using Blade-Gen proper co-ordinates in (r, z) direction and blade angle are inserted for each component. For further geometrical modification ANSYS Blade Editor is used and then geometries are transferred to ANSYS Turbo-Grid software for creation of mesh. The un-structured 3D mesh is also generated in ICEM CFD.

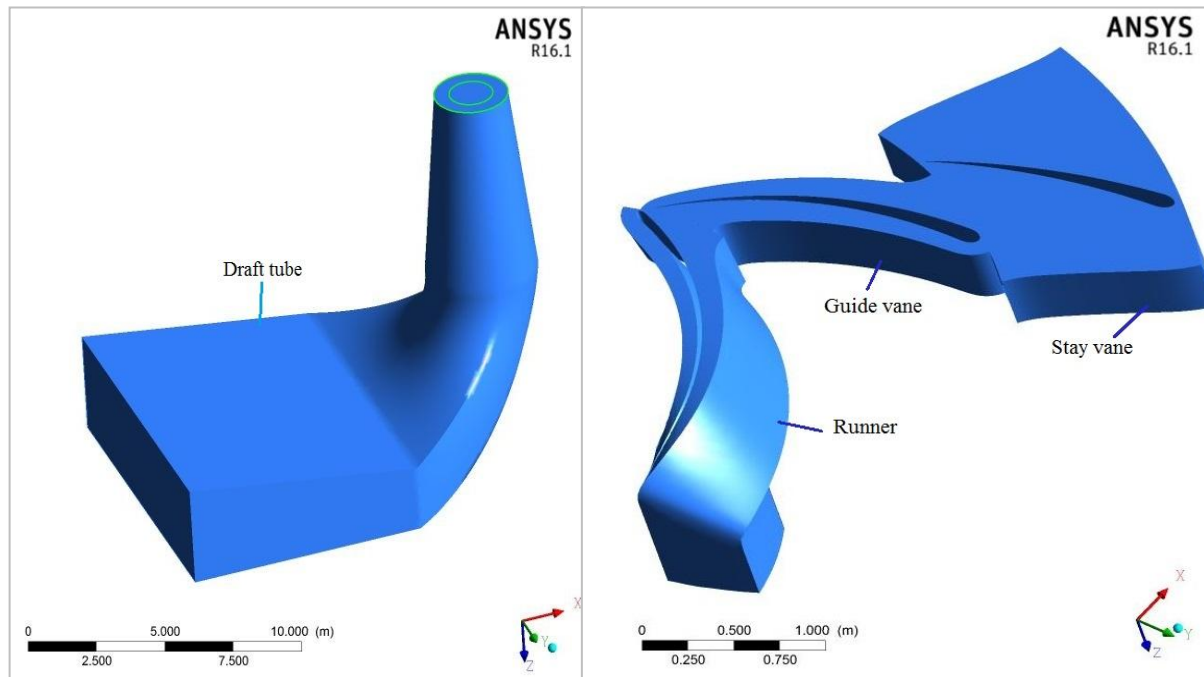


Figure 1:- Francis turbine assembled geometry

ANSYS Turbo-Grid generates high quality structured volumetric mesh of turbo-machineries. The 3D mesh can be generated for different turbine components with require number of layers along the blades. Total number of nodes for CFD analysis are huge and large computational memory is required, therefore one blade of runner, distributor and stay vanes are analyzed with rotational periodicity and complete draft tube is considered, but scroll casing with same boundary conditions is analyzed separately. The mesh is refined to get the required y^+ value for analysis. The mesh statistics is given in Table 2.

Table 2:- Hydraulic Francis turbine mesh statistics

Domain	Number of nodes	Number of element	Type of element
Runner	118910	104390	Hexahedra
Guide vane	326672	298056	Hexahedra
Stay vanes	659835	626640	Hexahedra
Draft tube	85204	474897	Tetrahedral
All domains	1190621	1503983	Tetra and hexahedra

Boundary Conditions and Simulation Setup:-

The inlet and outlet boundary conditions should be entered for CFD analysis and accuracy of results depend on location and nature of boundary conditions. Mass flow rate or discharge 68530 Kg is given at stay vanes as inlet boundary condition and 1 atmosphere static pressure at outlet of draft tube is specified as outlet boundary condition. The stay vane angle of 20 degree and five guide vane openings 34, 38, 42, 46, 50 and 54 degree from tangential direction are considered for simulation with the design discharge to obtain peak efficiency regime. The all components are stationery except runner which is rotating with a specified speed. All boundary surfaces are considered as smooth walls with no slip conditions.

Turbulence models based on Shear Stress Transport (SST) $\kappa-\omega$ is applied in this simulation for the viscous 3D flow analysis which is useful for catching the viscous sublayer.

Since there are very strong interaction between various components of hydraulic machineries especially between stay vanes, guide vanes, runner and draft tube of this Francis turbine. These interaction can be dealt with using some mixing models or interfaces. The following interfaces are used in this study.

- Rotational periodicity, is considered to simulate one blade of each domain.
- Frozen rotor, is applied at both sides of runner with guide vane and draft tube.
- Stage (Mixing plane), is used between stay vane and guide vane.

Results and Discussions:-

This Francis turbine was analyzed to assess its performance characteristics. The Francis turbine is simulated for different rotational speeds then the turbine is simulated for various guide vane angles to obtain the best guide vane where turbine reaches its high performance characteristics.

The speed of turbine which give high efficiency is searched to be 500 rpm where the discharge of 68530 kg is kept constant at guide vane angle of 38 degree. Efficiency variation with rotational speed is given in figure 2.

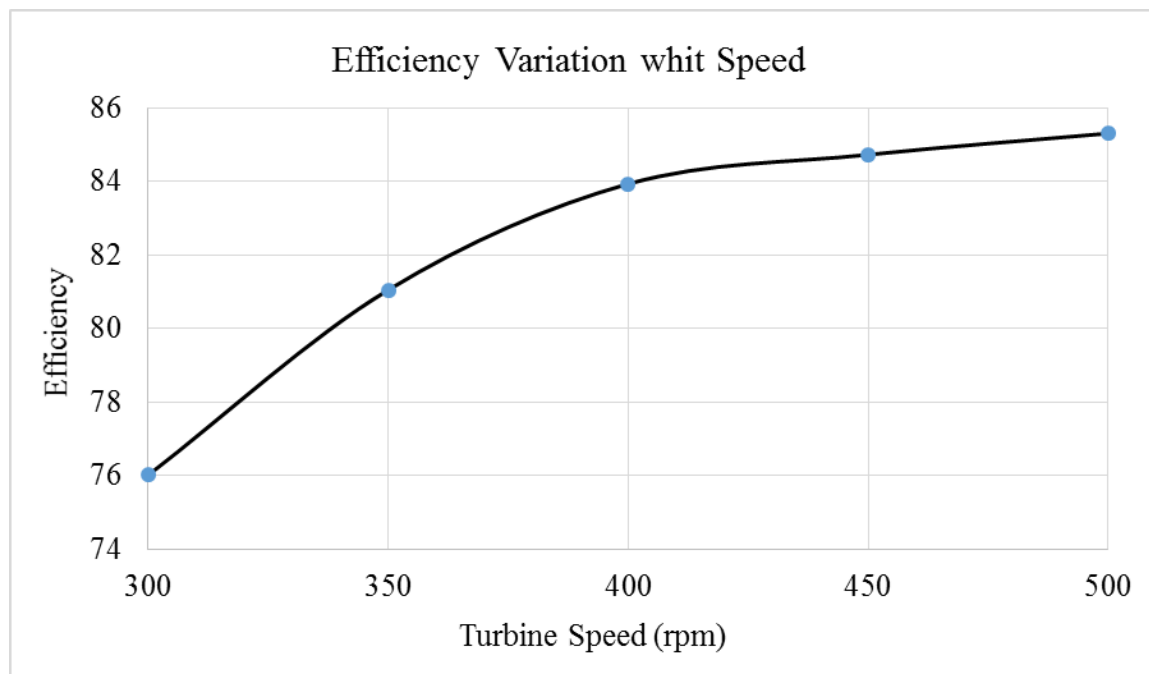


Figure 2:- Variation of efficiency with speed at constant discharge and guide vane angle of 38°

For understanding of efficiency behavior with guide vane angle the turbine is simulated for different guide vane angles at constant rotational speed of 500 rpm and discharge of 68530 kg. The guide vane angles are changed for each 4° and the best guide vane angle are obtained to be 50° and 54° where the efficiency is 88.089 and 88.085 respectively. Variation of efficiency with guide vane angle is given in figure 3.

The blade to blade pressure and velocity contours are shown in figure 4 with rotational speed of 500 rpm, guide vane angle of 50° and designed discharge. The pressure is seem to be maximum at stay vanes and minimum in runner and decreasing inside the guide vanes. Velocity is changing from minimum around 10 m/sec at stay vane to maximum of about 90 m/sec inside turbine runner.

Meridional flow path of turbine is included stay vane, guide vane and Francis runner from upper to down respectively. The meridional pressure and velocity variation are given in figure 5 and it seems that stay vane bear high pressure where turbine runner is dealing with high velocities.

Pressure contours on hub of stay vane, guide vane and turbine runner are shown in figure 6. It can be observed that pressure at stay vane is at some constant margins and it gradually decreasing in guide vane, but turbine runner is facing a large changing behavior of pressure.

The velocity vectors and velocity streamlines are given in figure 7. The velocity vectors indicates some shock loss at stay vane, guide vane and runner blades that has effects on the efficiency characteristics of turbine, but velocity streamlines are looked to be free of any vertices.

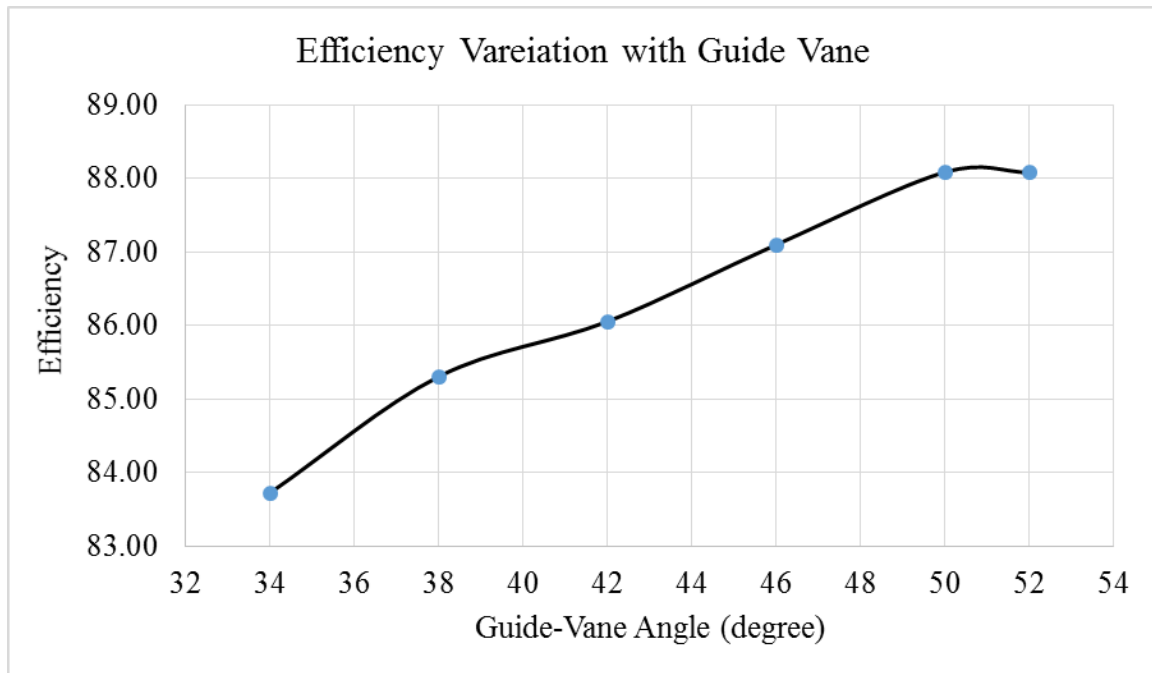


Figure 3:- Variation of efficiency with guide vane angle at speed of 500 rpm and constant discharge of 68530 Kg

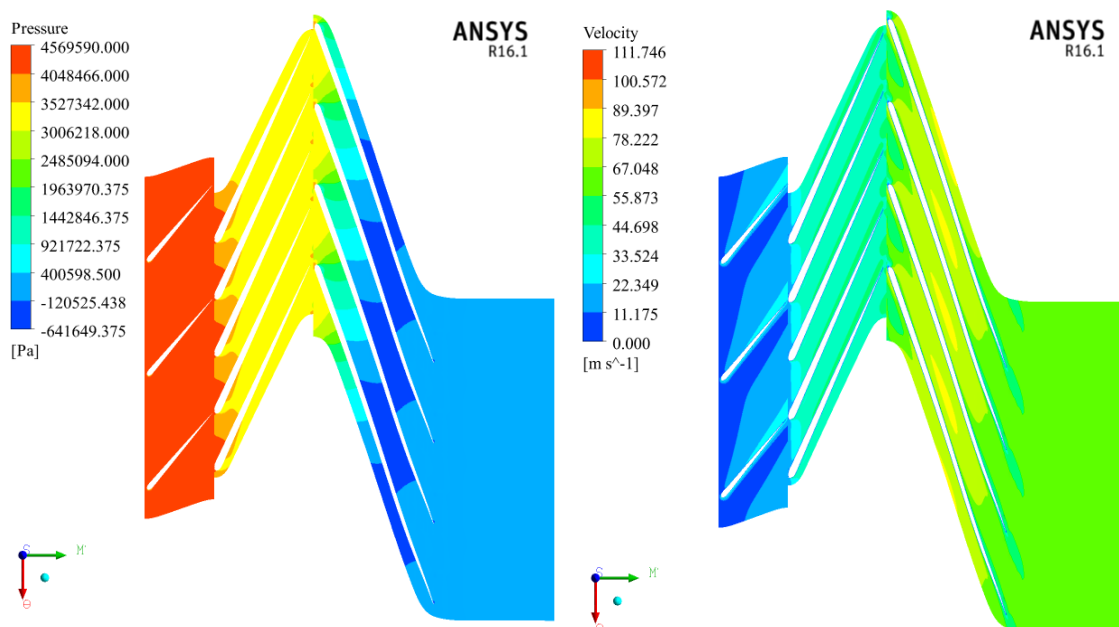


Figure 4:- Blade to blade pressure and velocity contours at 500 rpm speed and guide vane of 50°

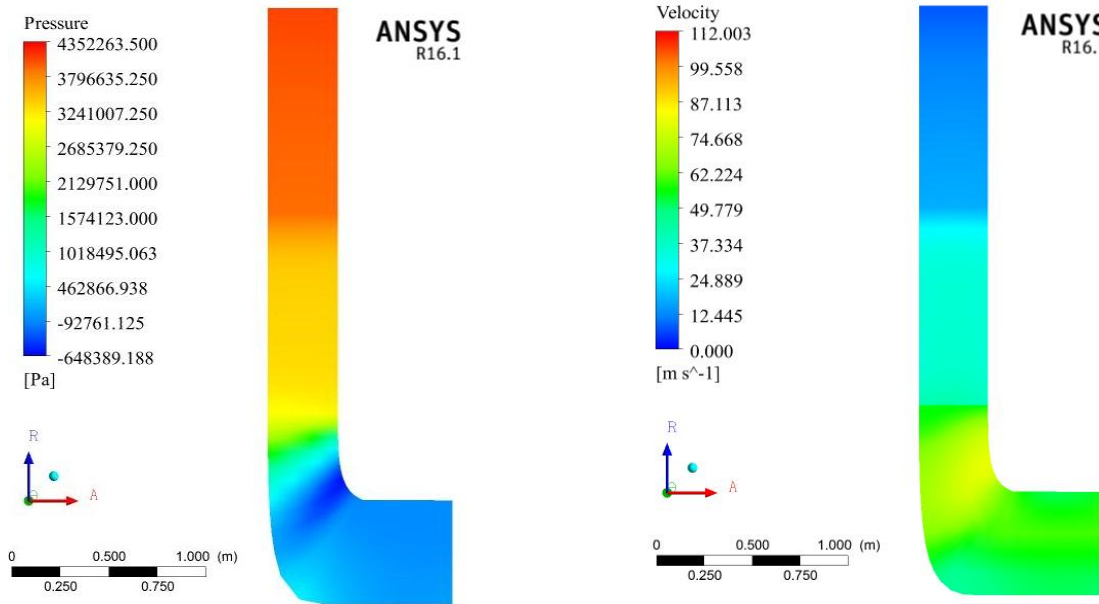


Figure 5:- Meridional pressure and velocity contours at speed of 500 rpm and guide vane of 50°

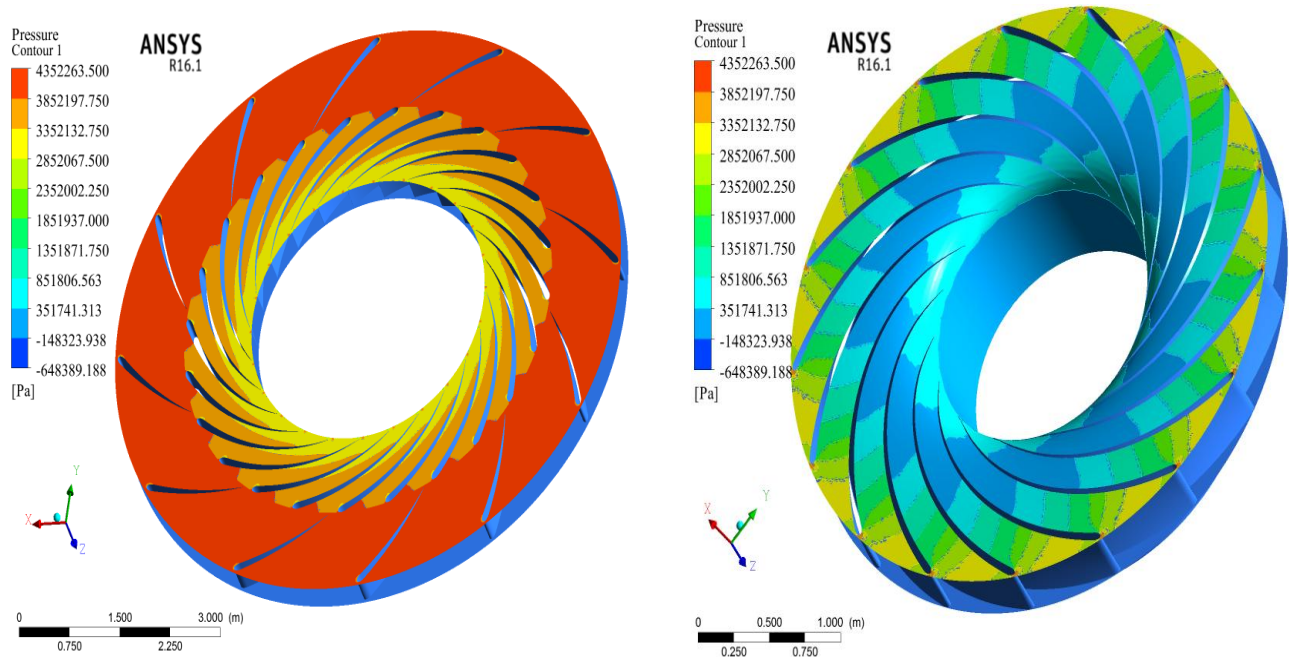


Figure 6:- Pressure contours on hub of runner, guide vanes and stay vanes at 500 rpm speed and guide vane of 50°

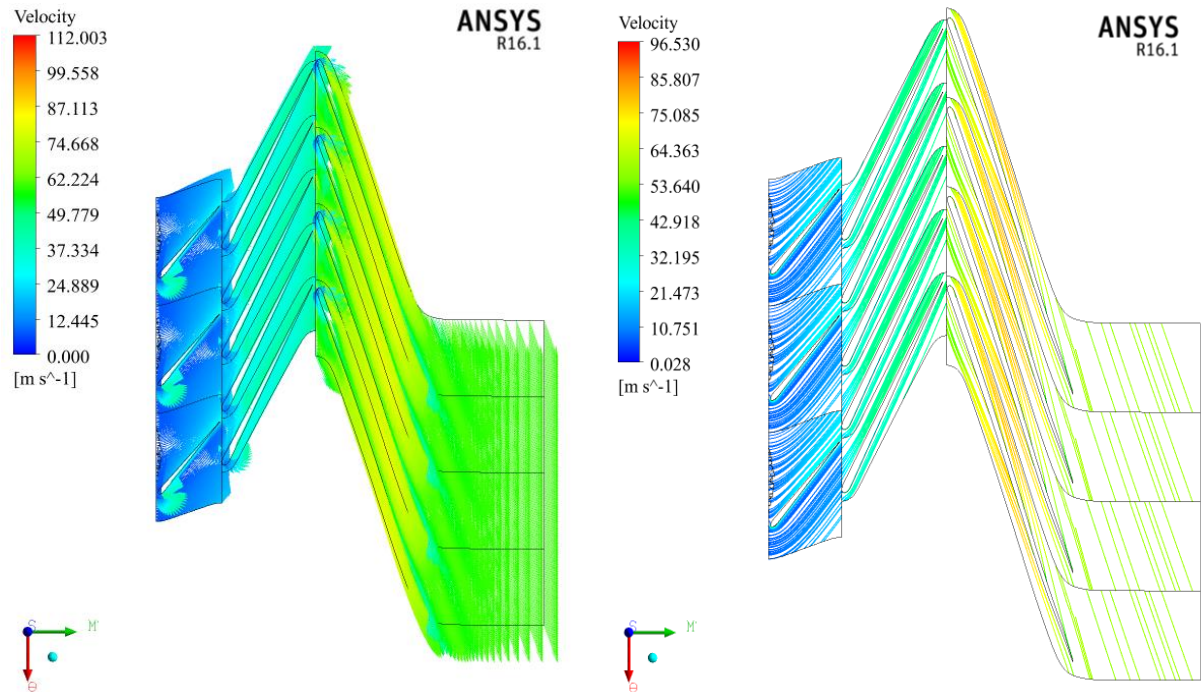


Figure 7:- Blade to blade velocity vectors and velocity streamlines at speed of 500 rpm and guide vane angle of 50°

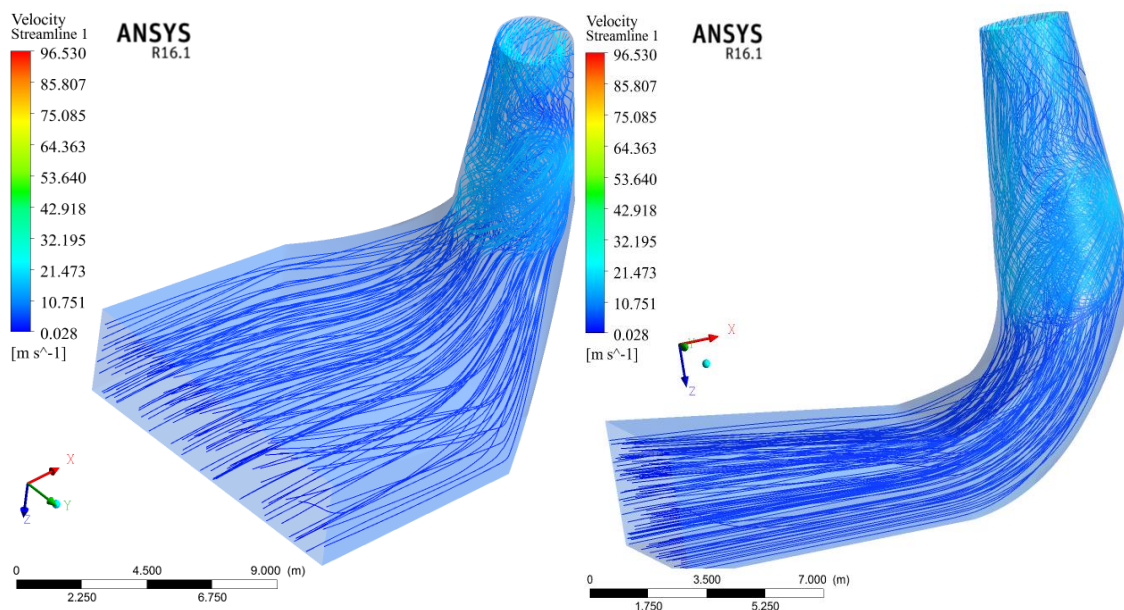


Figure 8:- Velocity streamlines in draft tube at 500 rpm speed and guide vane angle of 50°

The velocity streamlines are shown in figure 8 which are free of serious vortices and whirls at guide vane angle of 50° and rotational speed of 500 rpm. The efficiency draft tube is determined to be 76.568 which is maximum at guide vane angle of 50° and 500 rpm speed of turbine.

The normalized losses at turbine components, overall efficiency of turbine, hydraulic efficiency of simulated Francis runner and hydraulic efficiency of draft tube at different guide vane angles and 500 rpm speed at designed are given in Table 3.

The components of velocity triangles of simulated Francis turbine at different guide vane angles and 500 rpm speed with designed discharge are given in Table 4.

Table 3:- The normalized losses and efficiencies at runner speed of 500rpm

Components	GV = 34°	GV = 38°	GV = 42°	GV = 46°	GV = 50°	GV = 54°
Net head (m)	315.057	338.246	363.068	387.659	406.844	428.853
Runner loss (%)	11.770	11.229	9.245	9.421	9.673	8.974
Guide vane loss (%)	0.545	0.666	0.641	0.794	1.371	1.825
Stay vanes loss (%)	0.235	0.216	0.206	0.188	0.217	0.190
Draft tube loss (%)	2.246	2.003	1.922	1.727	1.560	1.529
Total loss (%)	14.796	14.114	12.014	12.130	12.821	12.518
Overall efficiency of turbine (%)	83.726	85.308	86.057	87.102	88.089	88.085
Hydraulic efficiency of runner (%)	80.963	82.143	84.696	84.383	86.159	86.477
Draft-tube efficiency (%)	71.251	71.696	68.853	71.227	76.568	76.245

Table 4:- Normalized velocity triangle's components of the Francis runner at different guide vane angel and speed of 500 rpm

Velocity components	GV = 34°	GV = 38°	GV = 42°	GV = 46°	GV = 50°	GV = 54°
Inlet velocity triangle						
Rotational velocity (u_1)	1.212	1.170	1.129	1.093	1.067	1.039
Velocity (W_1)	0.8869	0.8258	0.7652	0.7150	0.6711	0.6308
Velocity blade to blade (C_1)	0.8864	0.8253	0.7648	0.7146	0.6707	0.6305
Velocity meridional (C_{m1})	0.264	0.249	0.232	0.224	0.212	0.203
Velocity circumferential (C_{u1})	0.846	0.786	0.728	0.678	0.635	0.596
Flow angle (α_1)	18.610°	18.676	19.603	18.916	18.985	18.916
Runner blade angle (β_1)	35.807°	33.050	30.088	28.407	26.237	24.651
Outlet velocity triangle						
Rotational velocity (u_2)	0.693	0.669	0.651	0.626	0.611	0.595
Velocity (W_2)	0.7043	0.6827	0.6677	0.6398	0.6231	0.6063
Velocity blade to blade (C_2)	0.7040	0.6824	0.6673	0.6395	0.6228	0.6060
Velocity meridional (C_{m2})	0.278	0.267	0.250	0.248	0.242	0.237
Velocity circumferential (C_{u2})	0.637	0.618	0.610	0.580	0.565	0.549
Flow angle (α_2)	23.783	23.805	22.922	23.829	23.773	23.844
Runner blade angle (β_2)	78.704	79.144	80.725	79.430	79.113	79.078

Conclusions:-

The CFD analysis results shows similar pattern for velocity and pressure variation on turbine blades with variation of rotational speed but it will be effected with variation of guide vane angle. The distribution between hub and shroud, efficiency and power output affected by the rotational speed of the runner and guide vane angle. The maximum efficiency and output power occurs at the same rotational speed and guide vane angle. The computed loss is minimum in draft tube at point of maximum efficiency. The streamline and pressure contour plots in different component confirm with actual flow behavior in mixed flow turbine.

The best operating regime can be easily identified from computed flow parameters, losses and flow pattern from simulation results. Hence, it is concluded that using CFD approach is very effective and fast to study the performance and 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.

References:-

1. Vishnu Prasad (2012) "Numerical simulation for flow characteristics of axial flow hydraulic turbine runner", *Energy Procedia 14 (2012) 2060 – 2065*, Volume 4, Available online at www.sciencedirect.com
2. Vishnu Prasad, Manoj Kumar Shukla, Rajeev Jain, S. N. Shukla (2011) "CFD analysis of 3-D Flow for Francis Turbine" *MIT International Journal of Mechanical Engineering, Vol 1 No 2. Aug 2011. ISSN No. 2230-7699 MIT Publications.*

3. Vishnu Prasad, V K Gahlot and P Krishnamachar (2009) "CFD approach for design optimization and validation for axial flow hydraulic turbine" *Indian Journal of Engineering and Material Science*, Vol. 16, August 2009. PP. 229-236
4. Ruprecht, A., Heitele, M., Helmrich, T., Moser, W. and Aschenbrenner, T. (2014) "Numerical Simulation of a Complete Francis Turbine including unsteady rotor/stator interactions" Article is online Available on <https://www.researchgate.net/publication/267421304>
5. Maciej Kaniecki and Zbigniew Krzemianowski (2016) "CFD analysis of high speed Francis hydraulic turbines" *TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY*, No. 131, 2016, 111–120
6. J Nicolle and AM Girox, JF Morissete (2014) "CFD Configuration for hydraulic turbine startup" *27th IAHR Symposium on Hydraulic Machinery and systems*, IOP Conf. series: Earth and Environmental Science 22 (2014) 032021
7. Huimin Xiao and Bo Yu (2010) "Hydraulic Design of Water Turbine Based on the Computational Fluid Dynamic" *2010 International Conference on Electrical and Control Engineering*. www.google.com
8. Diaelhaq Khalifa, M.Sc. (2013) "SIMULATION OF AN AXIAL FLOW TURBINE RUNNER'S BLADES USING CFD" *1st Annual International Interdisciplinary Conference*, AIIC 2013, 24-26 April, Azores, Portuga.
9. Gagnon J.M. and Deschenes C. "Numerical Simulation of a Rotor-Stator Unsteady Interaction in a Propeller Turbine" Article is online available on www.google.com
10. Sergey Cherny, Denis Chirkov, Denis Bannikov, Vasily Lapin, Vladimir Skorospelov, Irina Eshkunova and Alexander Avdushenko (2010) "3D numerical simulation of transient processes in hydraulic turbines" *25th IAHR Symposium on Hydraulic Machinery and Systems*. September 20-24, 2010 Timisoara, Romania.
11. Gabriel Dan Ciocan, Monica Sanda Iliescu, Thi Cong Vu, Bernd Nennemann and Francois Avellan (2007) "Experimental Study and Numerical Simulation of the FLINDT Draft Tube Rotating Vortex" *Journal of Fluids Engineering*. Vol, 129, February 2007.
12. Sector Assessment (summary) "Energy Sector Road Map of Afghanistan" *Energy Supply Improvement Investment Program*. RRP AFG 47282-001.
13. ANSYS Technical Brief "Innovative Turbulence Modeling: SST Model in ANSYS CFX" www.ansys.com also article is available online on www.google.com
14. Arthur C. Huang, Edward M. Greitzer, Choon S. Tan, Eugene F. Clemens, Steven G. Gegg and Edward R. Tumer (2012) "BLADE LOADING EFFECTS ON AXIAL TURBINE TIP LEAKAGE VORTEX DYNAMICS AND LOSS" *ASME International*. <http://hdl.handle.net/1721.1/86369> and <http://creativecommons.org/licenses/by-nc-sa/4.0/>
15. H. Chen, M. Abidat, N. C. Baines and M. R. Firth (1992) "The Effects of Blade Loading in Radial and Mixed Flow Turbines" *The American Society of Mechanical Engineers*. 345 E. 47 St. New York, N. Y. 10017
16. Salim .M. Salim and S. C. Cheah (2009) "Wall y+ Strategy for Dealing with Wall-bounded Turbulent Flows" *The international Multi Conference of Engineers and Computer Scientists 2009*. Vol 2. IMECS 2009, March 18-20, 2009, Hong Kong
17. Mohd ARIFF, Salim M. SALIM and Siew Cheong CHEAH (2009) "Wally+ Approach for Dealing with Turbulent Flow over a Surface Mounted Cube: Part 2 – High Reynolds Number" *Seventh International Conference on CFD in the Minerals and Process Industries*. CSIRO, Melbourne, Australia 9-11 December 2009
18. Wahidullah Hakim Safi and Dr. Vishnu Prasad (2016) "GIS – BASED MORPHOMETRIC ANALYSIS OF KUNAR RIVER BASIN IN AFGHANISTAN" *International Journal of Advanced Research (IJAR)*. ISSN NO. 2320-5407. Int. J. Adv. Res. 4(12), 882-892. www.journalijar.com
19. Wahidullah Hakim Safi, Dr. Vishnu Prasad and Dr. Ruchi Khare (2017) "Simulation for Sediment assessment a case study of Kunar River in Afghanistan" *International Journal of Advanced Research (IJAR)*. ISSN: 2320-5407 Int. J. Adv. Res. 5(2), 406-416. www.journalijar.com
20. Santiago Lain, Manuel Garcia, Brian Quintero and Santiago Orrego (2010) "CFD Numerical simulations of Francis turbines" *Rev. Fac. Ing. Univ. Antioquia* NO. 51 pp. 24-33. February, 2010. Online available on www.google.com
21. Elias Mikael Vagn Siggeirsson and Steinn Gunnarsson "Conceptual design tool for radial turbine" Printed by Chalmers University of Technology Repro-service Gothenburg, Sweden 2015. A available online on www.google.com