iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** VI **Month of publication:** June 2017

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Analysis On Hydraulic Performance of Kaplan Turbine Based on Elbow Draft Tube Geometry

Lekha Mourya[1], Rahul Mishra[2], Swapnil Jain[3]
[1]Post graduate student of Thermal Engineering,
[2]Assistant Professor, [3]Associate Professor, Shri Shankracharya Engineering College (CG)

Abstract: ***This article focuses on the analysis on hydraulic performance of Kaplan Turbine based on Elbow draft tube geometry. A draft tube is a closed passage of gradually increasing cross sectional area which connects the runner exit to the tail race. It may be made of cast iron, steel or concrete. It must be air tight and under all condition of operations its lower end must be submerged below the level of water in the tail race. The draft tube recovers part of K.E. coming out of runner into useful pressure energy and thus improves the performance of turbine. The provision of draft tube at runner outlet decrease the pressure and lead to problem of cavitation which can be minimized by proper setting of runner above tail race. The performance of draft tube significantly affects the efficiency of a reaction turbine. For low heads and high flow rates, the draft tube losses are considerably very large (up to 50%). Therefore, the diffuser becomes the main part in the shape of draft tube. Also, it allows to install the turbine below the tail race without losing the head, and to direct the flow into the tail race. In addition, the draft tube is one of the most difficult parts to discuss on the behalf of flow parameters; this is because of the influence of many complex flow perspectives such as turbulence, unsteadiness, flow separation, swirl, curvature streamline and vortex breakdown. The effects from the above mentioned are very less and also they are improved by various design considerations.***

In the work presented here, the effect of change in geometry of the draft tube to the hydraulic efficiency of Kaplan turbine is evidently. For different guide vane openings and mass flow rates, in each case turbine with the circular draft tube gives the maximum efficiency while the turbine with the square draft tube having dividing pier gives the minimum efficiency.
Keywords: Draft Tube, Kaplan Turbine, Hydraulic Performance, hydro power, efficiency.

I. INTRODUCTION

The fossil fuels (Non-renewable resources of energy) emit greenhouse gases, which produces global warming and depletion of the ozone layer. The burning of fossil fuels causes a negative impact on the environment with pollution. Hydropower is the source of energy which is renewable and non-polluting with the operational life about 100 years and these are gaining global attention due to depleting of fossil fuels and their harmful environmental effect.

When the total energy of flowing water is converted into kinetic energy through nozzle and water jet move in the atmospheric and pressure does not change during flow through runner, such a turbine is called an impulse turbine. If only part of pressure energy is converted into kinetic energy in guide mechanism and the remaining part of pressure energy changes during the passage of the water through runner, the type of turbines are called reaction turbines. In reaction turbines, the flow through runner may be axial, radial or mixed.

Hydro power plants generate one fifth of the total electrical power produced in world. The small improvement in the hydrodynamic design and efficiency can contribute a great deal to the supply of the electric power. The efficiency of a hydropower plant depends on a number of parameters, such as: Turbine efficiency, Draft tube efficiency and Generator efficiency. Main component of reaction turbine (Kaplan turbine) are Spiral Casing, Stay Vanes, Guide Vanes, Runner and Draft Tube. The present study is upon draft tube. A draft tube is a closed passage of gradually increasing cross sectional area which connects the runner exit to the tail race. It may be made of cast iron, steel or concrete. It must be air tight and under all condition of operations its lower end must be submerged below the level of water in the tail race. The draft tube recovers part of K.E. coming out of runner into useful pressure energy and thus improves the performance of turbine. The provision of draft tube at runner outlet decrease the pressure and lead to problem of cavitation which can be minimized by proper setting of runner above tail race.

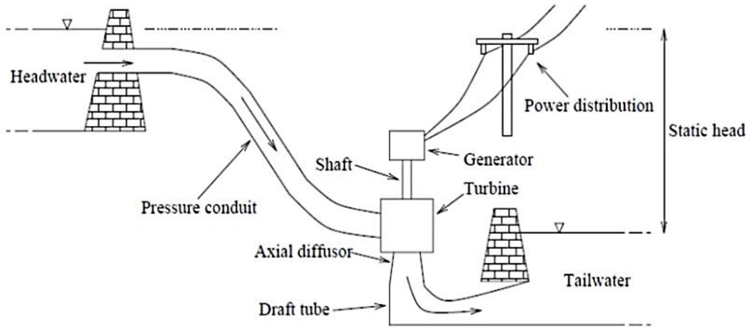

Fig. 1 Schematic diagram of hydro power plant

A. Main Components of Reaction Turbine

1) Spiral Casing, 2. Stay Vanes, 3. Guide Vanes, 4. Runner, 5. Draft Tube.

a) Draft Tube: A draft tube is a closed passage of gradually increasing cross sectional area which connects the runner exit to the tail race. It may be made of cast iron, steel or concrete. It must be air tight and under all condition of operations its lower end must be submerged below the level of water in the tail race.

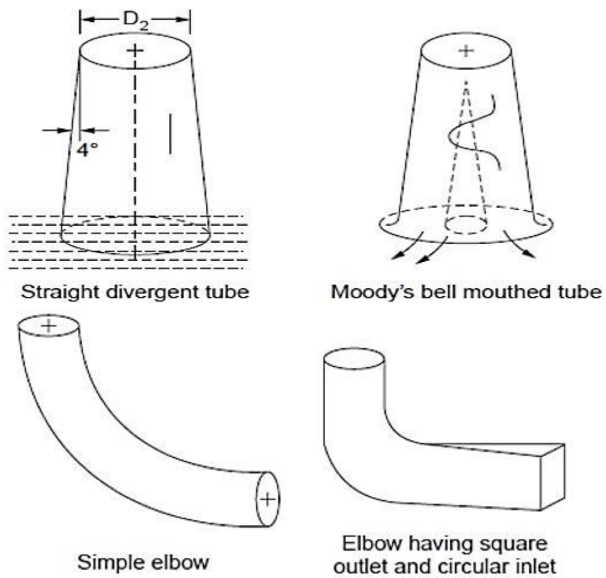

Fig. 2 Various Shapes of Draft Tubes.

II. LITERATURE REVIEW

Summary of some research papers are mentioned here.

A. Review on Development of Draft Tube

With the emergence of the reaction turbine draft tube comes in to consideration in the first half of the nineteenth century. The first draft tube was used by the Henschel and Jonval between 1837-1841. In the slow speed reaction turbines the draft tube affects very much on their power characteristics. But due to the transmission problems there was a need of small hydro power stations, to fulfil that requirements small turbines were designed. First experimental facilities were established in 1879.In 1903 a special hydraulic turbine laboratory was built in Zurich. Initially the draft tube had a cylindrical shape and were used only to connect the runner to trail race. These turbines were constant cross-sectional area helped only in the use of the static vacuum. i.e. were useful only for positive height of runner above the trail race. During 1909 to1929, numbers of investigations were carried out on straight diffuser. The work of A. Gibson, H. Hochschild and I. Nikurdze refers to this field. H.Hochschild and I. Nikurdze observed the coefficient of resistance and characteristics of flow. Later in USA the idea of bell mouthed draft tube in the nineteenth century comes in

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

consideration but its practical application found latter. In 1917, W.M. White designed and built a bell mouthed tube for the hydraulic stations at Niagara Falls. In 1919, moody suggested that the well mouthed tube with a cone that would fill the dead zone created in the swirling flow. The moody draft tube was widely used in large hydroelectric stations in USA. Numbers of investigations were carried out in the USSR under typical operating conditions of turbine. A large number of research laboratories, research institutes and higher technical institutions in USSR are engaged in experimental investigations of such problems. The hydraulic turbine laboratory of LAZ (named after the XXII congress of the CPSU), the chair of hydraulics and hydraulic machines of the MVTU, the laboratory of hydraulic machines of VNII Givromash, LPI, VNIIG, NIS Giroproekt, the MISI and other large number of laboratories and investigators are engaged to the improvements in the design of draft tube.

B. Flow simulations of draft tube

Solnishkov V.A. et al. [1973] investigated the influence of length of elbow on the power characteristics of the turbine on models of draft tube of height 1.915 D1.Varying the length of elbow from 1.75 to 2.3 D1 shows that maximum efficiency is obtained for length of elbow equal to 6.0 D1. Wahl Tony Lee et al. [1990] presented the phenomenon of draft tube surging using a model hydraulic turbine. The Pressure fluctuations at the throat of the draft tube were measured using pressure transducers, and the signals were analyzed using a dynamic signal analyzer and a digitizing oscilloscope. Dimensionless pressure and frequency parameters were calculated for the dominant pressure pulsation at each test point. The dimensionless parameters have been related to the dimensionless swirl parameter of the flow in the draft tube, and maps have been constructed showing the variation of these parameters on the turbine hill curve. Ernesto Casartelli et al. [2006] has done numerical simulation on Francis turbine and the flow analysis showed that the interaction of the main runner-outflow and the leakage flow has little influence on the overall runner performance. The runner-outflow, however, is affected by the leakage flow, especially the meridional and the tangential velocity components along the cone walls. These effects do not disappear due to the mixing with the main runner-outflow and alter therefore the inlet condition to the draft tube significantly. The draft tube flow is computed for various operating points from part load to overload. The results show that the draft-tube pressure recovery is sensitive to the different distributions of meridional and tangential velocity component at the runner outlet. The recirculation zone at the inner side of the elbow is influenced by the leakage flow.

III. PROBLEM IDENTIFICATION

A. Formulation of problem

In order to decrease the cost and get better efficiency, numerical simulation is widely used for fluid mechanics process and becoming a very important method for the design, optimization and development of new products. With the rapid development of computer technology, the computer numerical simulation technology obtained wide application in fluid machinery part. There are many researches about finite element method (FEM) simulation on the elbow draft tube for geometrical analysis. But most of these researches took place in laboratory, lacking of combination of practical process. Therefore, the finite element analysis software CFD process is used to simulate the process.

B. The outcome of the literature reviews

A number of research papers have been within on the CFD application to turbo machine. Most of the work is on steady flow simulations. The detailed steady of the performance characteristics of draft tube under unsteady flow simulation is still a thrust area.

IV. METHODOLOGY AND GOVERNING EQUATIONS

A. General

The performance of draft tube significantly affects the efficiency of a reaction turbine. For low heads and high flow rates, the draft tube losses are considerably very large (up to 50%) as seen in fig. 3. The purpose of draft tube is to recover the kinetic energy (velocity) into pressure energy, which would be pure losses if draft tube not installed. Therefore, the diffuser becomes the main part in the shape of draft tube. Also, it allows to install the turbine below the tail race without losing the head, and to direct the flow into the tail race. In addition, the draft tube is one of the most difficult parts to discuss on the behalf of flow parameters; this is because of the influence of many complex flow perspectives such as turbulence, unsteadiness, flow separation, swirl, curvature streamline and vortex breakdown.

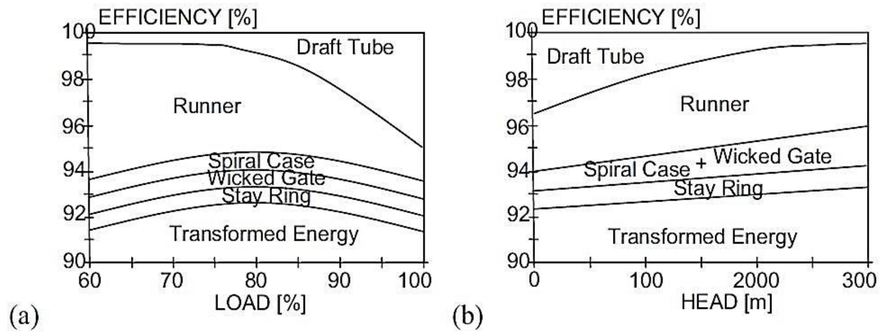

Fig. 3 Typical losses in of a reaction turbine (a) efficiency-load (b) efficiency-head.

B. *Principles of Draft Tube*

The principle of draft tube is determined by the use of Bernoulli's Equation between section inlet 1-1 and outlet 2-2, Fig 4.

$$\frac{p_1}{\rho g} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{V_2^2}{2g} + h_f. \qquad \text{(i)}$$

where p is the absolute pressure, z is the height, V is the mean velocity and h_f is the hydraulic losses in the draft tube. The absolute pressure at section 2-2 can also be defined as

$$p_2/\rho g = z_2 + p_{atm}/\rho g, \qquad \text{(ii)}$$

where p_{atm} is the atmospheric pressure. Assuming that the turbine is installed at height, H_s which is approximately equal to z_1, hence equation (i) becomes,

$$\frac{p_1}{\rho g} = \frac{p_{atm}}{\rho g} - \left(H_s + \left(\frac{V_1^2}{2g} - \frac{V_2^2}{2g} - h_f\right)\right). \qquad (1)$$

The equation 1 shows that the draft tube generates a low pressure region under the runner, which can be utilised by the turbine. There are two terms associated with this lower pressure; static fall of pressure and dynamic fall of pressure, H_s and $V_1^2/2g - V_2^2/2g - h_f$ respectively.

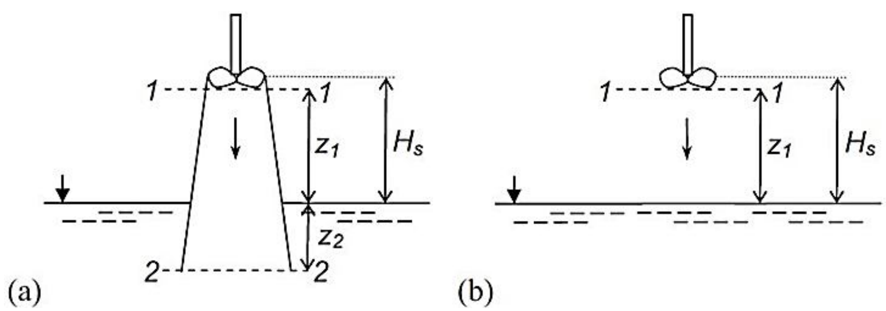

Fig. 4 Hydraulic principles of draft tube (a) with; (b) without.

C. *Draft Tube Recovery and Efficiency*

In actual design of Hydro Electric Stations, it is rarely possible to use draft tubes recommended by turbine designers without changing some dimensions. Because of the diffusion of the flow in the draft tube after the runner an additional dynamic regain is created and this is equal to,

$$\Delta h_d = \frac{v_3^2 - v_5^2}{2g} - h_{loss} \qquad (2)$$

Where, v_3 = Inlet velocity after the runner (m/s)

v_5 = Outlet velocity at exit of draft tube (m/s)

The magnitude of this regain depends not only on through flow & the geometry of the inlet and outlet sections F_3 and F_5 but also on the flow characteristics at both the sections which are governed by operating conditions of the turbine and the quality of draft tube. Measuring the hydraulic quality of draft tubes in terms of its recovery co-efficient

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

$$\eta_d = \frac{2g\Delta h_d}{v_3^2} \times 100 \quad (3)$$

Where, Δh_d = Head recovery (m)

D. Head Loss and Relative Loss in Draft Tube

Measuring the head loss in draft tubes is given by:

$$h_{loss} = \frac{P_{o3} - P_{o5}}{\gamma} \quad (4)$$

Measuring the relative loss in draft tubes is given by:

$$h_{rld} = \frac{2gh_{loss}}{v_3^2} \times 100 \quad (5)$$

E. Computation of Flow Parameter

The following parameters are to be computed using the observed data from experimental test of turbine model:

Net head: $H_n = \frac{TP_{CI} - TP_{DO}}{\gamma}$ (6)

Head utilized by the runner: $H_R = \frac{2\pi NT}{60 * \gamma Q}$ (7)

Average flow velocity at inlet: $C_1 = \frac{Q}{A_1}$ (8)

Average flow velocity at outlet i.e. in draft tube: $C_2 = \frac{Q}{A_2}$ (9)

Net head on turbine: $H_n = \left(\frac{P_1}{\gamma} + \frac{C_1^2}{2g} + z_1\right) - \left(\frac{P_2}{\gamma} + \frac{C_2^2}{2g} + z_2\right)$ (10)

Unit speed: $n_{11} = \frac{nD}{\sqrt{H_n}}$ (11)

Unit discharge: $Q_{11} = \frac{Q}{D^2\sqrt{H_n}}$ (12)

Input power: $P_{in} = \gamma Q H_n$ (13)

Output power: $P_{out} = \frac{2\pi nT}{60}$ (14)

Hydraulic efficiency: $\eta_H = \frac{P_{out}}{P_{in}} * 100$ (15)

Speed factor: $SF = \frac{nD}{\sqrt{gH_n}}$ (16)

Discharge factor: $DF = \frac{Q}{D^2\sqrt{gH_n}}$ (17)

F. Application of CFD

CFD used for this project. CFD is a computational technology that enables to study the dynamics of matters that flows. CFD is predicting what will happen, quantitatively, when fluids flow even with the complications of simultaneous flow of heat, mass transfer, phase change, chemical reaction, mechanical movement, stresses in and displacement of immersed or surrounding solids. CFD include expressions for the conservation of mass, momentum, pressure, species and turbulence.

V. RESULT

The performance analysis of existing Kaplan turbine has been carried out using full-3D turbulent flow approach in the annulus space of model consisting of spiral casing, stay vanes, guide vanes, runner and draft tube for an operating condition. The flow area is affected by changing the different geometries of draft tube designed. The variation of computed flow parameters are presented in the graphical form. Result is elaborated in different headings below:

A. Common Input Parameters of Turbine

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

The numerical simulation for any flow domain requires 3-D geometry of flow space, boundary conditions, nature of flow and properties of fluid. Some parameters are required to be specified for the numerical simulation depending upon potential or viscous flow analysis. The common parameters with their values used during analysis are given below:

Type of fluid	Water
Density of water	1000 Kg/m^3
Specific weight of water	9810 N/m^3
Kinematic viscosity of water	10^{-6} m^2/s
Boundary wall	smooth with no slip
Input boundary condition	mass flow rate specified as
	0.525 m^3/s for 35º guide vane opening
	0.620 m^3/s for 40º guide vane opening
	0.714 m^3/s for 50º guide vane opening
Outlet boundary condition	specification of reference pressure at draft tube outlet as 0 atm
Stationary blade rows	stay ring and guide vanes
Rotating blade row	runner with rotational speed specified as
	1050 for 35º guide vane opening
	1150 for 40º guide vane opening
	1375 for 50º guide vane opening
Type of interfaces	Fluid-Fluid
Frame change/Mixing model	Casing and stay vanes – None
	Stay vane and guide vane – None
	Guide vane and runner – Frozen Rotor
	Runner and draft tube –Frozen Rotor
Interface model	General connection
Pitch change	Automatic, GGI Connection
Turbulence model	SST κ-ω model
Flow type	Incompressible

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Table 1: Abbreviation used for types of draft tube used

Type of draft tube	Abbreviation used
Rectangular Draft tube	rdt
Rectangular with curved edges Draft tube	rcdt
Rectangular with dividing pier Draft tube	rddt
Square Draft tube	sdt
Square with curved edges Draft tube	scdt
Square with dividing pier Draft tube	sddt
Elliptical Draft tube	edt
Circular Draft tube	cdt

B. Validation of Simulation Results with Experimental Data: The results are shown in fig. 5 for the variation of efficiency with respect to different loading conditions. It is found that the pattern of calculated efficiency from simulation and experimental results are nearly identical. The maximum efficiency for rated condition i.e. for 40º guide vane opening is 90.3% with an error of 1.8% with experimental data which is acceptable for CFD. The curve is parabolic and it shows that the efficiency is increasing from part load to rated load and then decreases.

Table 2– Comparison of Simulation results with the experimental data

Loading conditions	Simulation Results	Experimental results By researcher.	% of error
Part load (35º guide vane opening)	89.2	91.5	2.5%
Rated (40º guide vane opening)	90.3	92.0	1.8%
Over load (50º guide vane opening)	88.5	90.8	2.5%

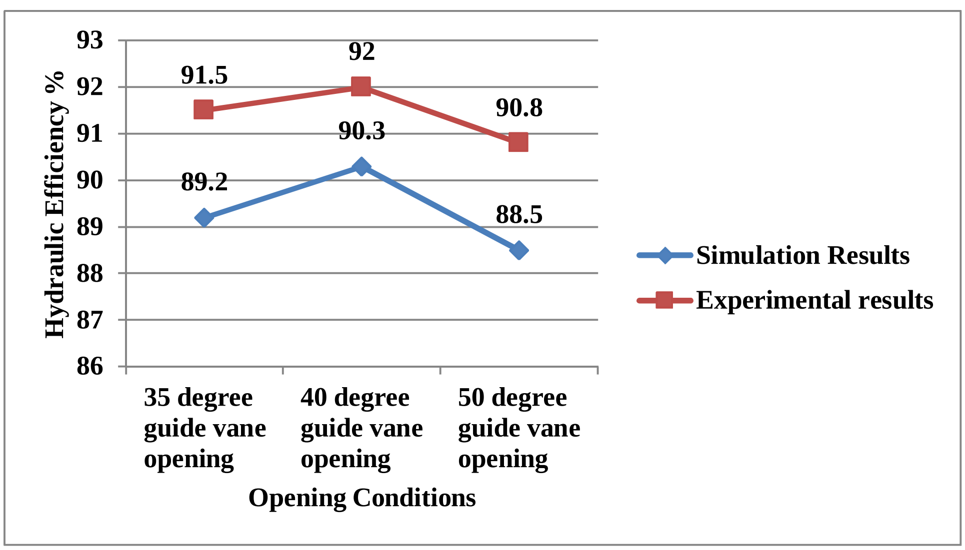

Fig. 5 Comparison of efficiencies with respect to loading conditions.

C. Effect of Draft Tube Geometry on its Performance

The effect of change in geometry of the draft tube to the hydraulic efficiency of turbine is evidently shown in fig. 6.2. For different guide vane openings and mass flow rates, in each case turbine with the circular draft tube gives the maximum efficiency while the

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

turbine with the square draft tube having dividing pier gives the minimum efficiency.

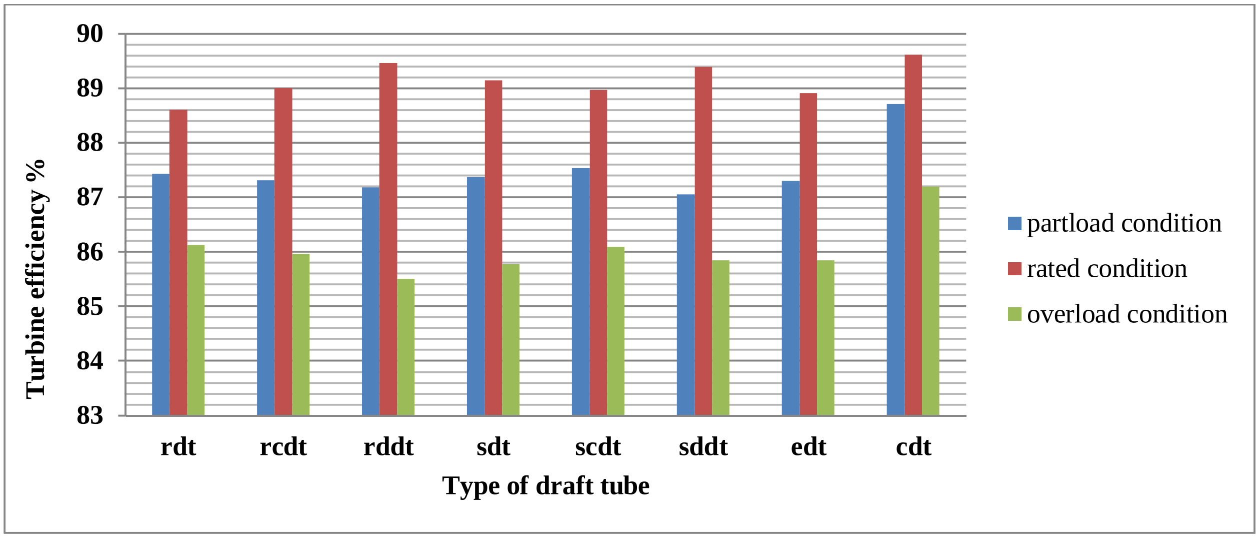

Fig. 6 Effect of loading conditions on different type of draft tube

D. Table 3: Table Comparison of computed parameters for draft tube

For 35 degree guide vane opening

Type of Draft tube	D.T. Head Loss (m)	D. T. Net Head Recovery (m)	Draft tube efficiency
Rdt	0.000285036	1.188	94.756
Rcdt	0.000282853	1.178	94.574
Rddt	0.000300801	1.213	94.806
Sdt	0.000277579	1.207	95.678
Scdt	0.000274625	1.181	95.436
Sddt	0.000292575	1.198	95.345
Edt	0.000285278	1.168	93.546
Cdt	0.000177033	1.212	95.559

For 40 degree guide vane opening

Type of Draft tube	D.T. Head Loss (m)	D. T. Net Head Recovery (m)	Draft tube efficiency
rdt	0.000602049	1.734	94.94
rcdt	0.000594001	1.721	94.78
rddt	0.000563209	1.756	94.96
sdt	0.000592733	1.762	95.85
scdt	0.000589524	1.722	95.62

sddt	0.00061447	1.751	95.67
edt	0.000597217	1.707	93.79
cdt	0.000463568	1.834	96.15

For 50 degree guide vane opening

Type of Draft tube	D.T. Head Loss (m)	D. T. Net Head Recovery (m)	Draft tube efficiency
rdt	0.000644978	2.225	94.79
rcdt	0.000639945	2.208	94.63
rddt	0.000689085	2.273	94.88
sdt	0.000616712	2.243	94.92
scdt	0.00061623	2.211	95.48
sddt	0.000643017	2.244	95.41
edt	0.000647634	2.191	93.61
cdt	0.000411865	2.273	95.62

VI. CONCLUSION

The effect of change in geometry of the draft tube to the hydraulic efficiency of Kaplan turbine has been analysed and evidently shown in Table 3. For different guide vane openings and mass flow rates, in each case turbine with the circular draft tube gives the maximum efficiency while the turbine with the square draft tube having dividing pier gives the minimum efficiency. From the research and analysing various types of draft tubes, we concluded that maximum efficiency achieved in Circular Draft Tube at 40° guide vane operating. It is found that the pattern of calculated efficiency from simulation and experimental results are nearly identical hence simulation result has been validated from experimental data. Redesigning the existing draft tube can be done by changing the shapes of draft tube, such as elliptical, square, circular and rectangular. Performance & efficiency of Kaplan turbine can be improved. We can apply this analysis to other turbines also so that the performance & efficiency can be improved.

VII. FUTURE SCOPE

Analysis can be done for effect of velocity components on the performance of draft tube. Unsteady flow simulation can be performed for different types of draft tubes working under similar conditions to find out most efficient among them.

VIII. ACKNOWLEDGEMENT

I take this opportunity to thank my teachers and my guide who provided constant encouragement and made it possible for me to take up challenge doing this project. I am grateful to Associate Prof. Swapnil Jain and Assistant. Prof. Rahul Mishra for their technical support, valuable guidance, encouragement and consistent help. Last but not least, I am thankful to entire staff of Mechanical department for their timely help and guidance at various stages of this project.

REFERENCES

[1] M.F. Gubin, "Draft Tubes of Hydro-Electric Stations", (1973), Amarind Publishing Company Private Ltd, New Delhi.
[2] Tony Lee Wah, (1990), "Draft Tube Surging Hydraulic Model Study", Colorado State University Fort Collins, Colorado.
[3] P.N. Modi S.M. Seth, (1991) "Hydraulic and Fluid Mechanics", Standard Book House, New Delhi.
[4] S.Mauri, J.L. Kueny, (2000), "Numerical Prediction of the Flow in a Draft Tube".
[5] B.D. Marjavaara, T.S. Lundstrom, (2003),"Automatic Shape Optimisation of a Hydropower Draft Tube", Joint Fluids Engineering Conference Honolulu, Hawaii, USA.
[6] Jorge Arpe, (2002) "Pressure Wall Measurement in the Whole Draft Tube: Steady and Unsteady Analysis", XXI IAHR Symposium on Hydraulic Machinery

and System Lausanne.
[7] R.K. Bansal, (2005)"a Textbook of Fluid Mechanics and Hydraulic Machines", Luxmi Publication, New Delhi.
[8] Romeo Susan-Resiga, Sebastian Muntean, Gabriel Dan Ciocan, Bernd Nennemann,(2006), "Jet Control of The Draft Tube Vortex Rope in Francis Turbines", 23rd IAHR Symposium – YokohamaVol.14.
[9] Tube Effect" IAHR Symposium on Hydraulic Machinery and Systems IOP Publishing.
[10] Nicoleta Octavia Tanase, Florentina Bunea, (2012), Gabriela Dan Ciocan, 'Numerical Simulation of the Flow in the Draft Tube of the Kaplan Turbine' U.P.B. Sci. Bull., Series D, Vol. 74, Issn 1454.
[11] Choi, Hyen-Jun, Mohammed Asid, Hyoung-Woon, Pil-Su, Sueg-Young, Young-Ho, (2013), "CFD Validation of Performance Improvement of a 500 kW Francis Turbine", Renewable Energy, Vol.54, pp. 111-123.
[12] http://www.British-Hydro.Org/Mini-Hydro/Images/Spiral-Case-Francis.Gif
[13] http://www.Hydrotech-Eng.Com/Equipment/Turbines/Francis-Turbines/
[14] www.frenchriverland.com.
[15] www.british-hydro.org/mini-hydro/images/spiral-case-francis.gif.
[16] www.hydrotech-eng.com/equipment/turbine/francis-turbine.
[17] www.nptel.ac.in