IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 13, Issue 4 Ver. III (Jul. - Aug. 2016), PP 54-59
www.iosrjournals.org

Design and Performance Analysis of Straflo Turbine Using CFD

Ajay Singh[1], Vishnu Prasad[2]
[1](*M. Tech. Scholar, Hydro Power Engineering, M.A. National Institute of Technology, Bhopal, India)*
[2](*Professor, Civil Engineering, M.A. National Institute of Technology, Bhopal, India)*

Abstract: *The distributor, runner and draft tube of a turbine are designed for a fixed head and discharge but in most of power stations, turbine operates at off-design values of head and discharge. The change in head and discharge lead change in local flow parameter like velocity and angles and this affect the performance of turbine. This makes customary to assess detailed flow behaviour and performance of turbine for off design operating conditions for efficient design. In present paper, a Straflo axial flow turbine has been designed for micro hydro power station and its performance parameters have been computed using commercial code Ansys CFX. The variation of performance parameters obtained from simulations are found to be very similar to an axial turbine characteristics. The best efficiency is achieved at design parameter values.*
Keywords: *runner, specific speed, SST model, Straflo turbine.*

I. Introduction

There is need to increase the use of renewable sources of energy over other sources like fossil and nuclear energy sources for sustainable development. Out of available renewable sources of energy, hydro energy is having least operating and maintenance cost after its installation. Around one-fifth of the world power requirement is fulfilled by hydropower [1]. Micro, mini and small hydro plants play a key role of rural electrification in many countries and they have greater capacity than all other renewable energy sources to make an instant impact on the replacement of fossil fuels. The water is conveyed mostly in pipes and on sloping ground and thus it gains additional energy due to falling slope which can be utilised for power generation.

The turbine is an important part in a hydro power plant because hydro energy is extracted from water and is converted into mechanical energy in turbine. The Straflo axial flow turbine is high specific speed fully tubular machine and used for low head and high discharge. Straflo turbines comprises turbine and generator into single unit are the best choice for micro hydro power development from water flowing in pipes. The generator is mounted around the periphery of runner. The blades of axial flow turbines are highly twisted [2] due to which there occurs change in angular momentum which forces the rotor to rotate along with generator shaft which in turn generates electricity. It is costly and time consuming to check turbine performance by experimental testing. Comparatively, CFD is an efficient tool to study flow behaviour inside turbine space providing flow details, component wise, in terms of both local and global parameters. CFD had been used by many investigators to carry out flow simulations for axial flow turbines to analyse their performance, pressure pulsation prediction, swirling flow investigation etc. [2, 3 and 4] and validated. In this paper, distributor, runner and a conical draft tube are designed for a low head horizontal axial flow Straflo turbine. The runner and distributor of Straflo turbine are designed in Ansys Bladegen and numerical simulations are carried out for turbulent flow in complete turbine space from distributor inlet to draft tube outlet by using SST turbulence model in Ansys CFX codes for different operating conditions of discharge and rotational speed. The velocity and pressure distribution from simulation results are used to compute axial flow turbine characteristics parameters and presented in graphical and tabular form.

1.1. Nomenclature

- g Acceleration due to gravity (m/s^2)
- H Net head on turbine (m)
- D Runner diameter (m)
- Q Discharge through turbine (m^3/s)
- N Rotational speed of runner (rpm)
- P_{01} Total pressure at runner inlet (Pa)
- P_{02} Total pressure at runner outlet (Pa)
- p_1 Static pressure at runner inlet (Pa)
- p_2 Static pressure at runner outlet (Pa)

II. Geometric Modelling

The Straflo turbine is designed for a discharge of 550 kg/s, a head of 7.5 m and a power output of 35 KW. The axi-symmetric axial distributor with fixed 7 vanes, propeller runner with 4 fixed blades and conical draft tube are designed. The computed values of inlet and outlet blade angles are used in Bladegen to generate blades. A suitable stagger angle variation is taken from hub to shroud so as to get blades with smooth curvature. The inlet flow angle for runner is taken as blade outlet angle for guide vanes. Axial distributor with twisted blades is used ahead of the runner. The three components are modelled separately with only a single blade for both the runner and distributor. Tetra/mixed mesh using robust (octree) method is used for meshing in Ansys ICEM CFD for whole of the turbine space separately for all components. The simulations have been carried out for different mesh sizes to check mesh dependency of flow parameters and to get Y+ value with acceptable limits. The distributor, runner and draft tube are modelled and meshed separately and connected through proper interfaces. The geometry of the turbine model used for simulations is shown in Fig.1.

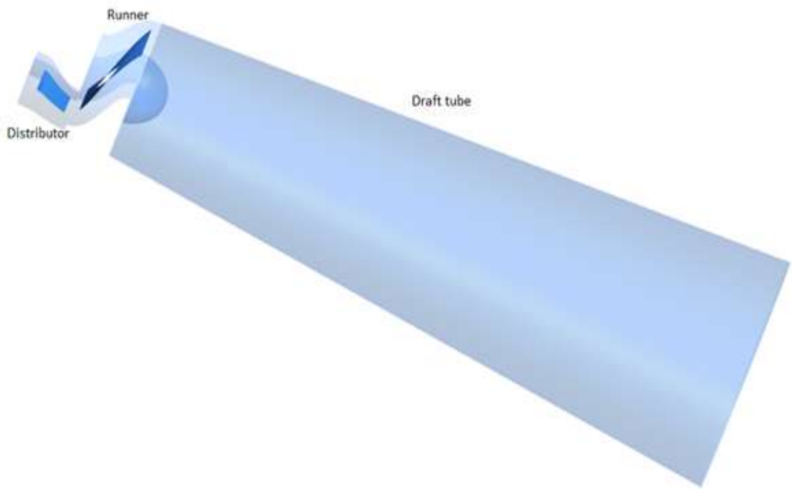

Fig.1 3D geometry of straflo turbine model for flow simulations

III. Boundary Conditions

The flow simulations have been carried out for three different mass flow rates specified at inlet of distributor and three different rotational speed of runner, including design and off design conditions. Smooth walls with no slip are chosen as boundary condition for blades, hub, shroud, draft tube cone. Reference pressure is taken as zero atmosphere. Periodicity is applied at runner and distributor because of the limited computational power**.** Distributor and draft tube are taken as stationary domain and runner as rotating domain. Frozen rotor type of interface is used for interfacing between stationary and rotating domains. The SST turbulence model is used in simulation for the flow inside turbine being complex and rotating.

IV. Formulae Used

The characteristic parameters are computed in dimensionless form using following formulae:

Specific energy coefficient $$\psi = \frac{gHD^4}{Q^2} \tag{1}$$

Speed factor $$SF = \frac{ND}{\sqrt{gH}} \tag{2}$$

Discharge factor $$DF = \frac{Q}{D^2\sqrt{gH}} \tag{3}$$

Runner head $$H_R = \frac{(P_{01}-P_{02})_{S\tan.frame} - (P_{01}-P_{02})_{Rot.frame}}{\rho g} \tag{4}$$

Hydraulic efficiency (%) $$\eta_h = \frac{H_R}{H} * 100 \tag{5}$$

Pressure coefficient $c_p = \dfrac{p - p_2}{\dfrac{\rho w_2^2}{2}}$ (6)

Hydraulic losses (%) $H_L = \dfrac{(H - H_R)}{H} * 100$ (7)

Degree of reaction $R = \dfrac{W_2^2 - W_1^2}{2gH}$ (8)

V. Results and Discussions

The grid dependency of different flow parameters is checked by performing numerical simulations for a number of tetrahedral elements ranging from 570637 to 1326245 at the design operating conditions. In hydraulic efficiency, a minor change of an order 0.08% was found after 800000 tetrahedral elements and thus all the numerical simulations are carried out for 795223 number of elements in the present paper. The flow simulation is done for discharge values varying from 500 kg/s to 600 kg/s and for rotational speed varying from 850 rpm to 950 rpm at the intervals of 50 kg/s and 50 rpm, respectively. Accordingly, speed factor and discharge factor are found to vary from 32 to 67 and 0.3 to 0.55, respectively. The variation of discharge factor with speed factor for varying discharge at three different runner speeds is shown in Fig.2. The net head increases as discharge increases at constant rotational speed and due to this both speed factor and discharge factor decrease. The decrease in discharge factor is due to higher increase in net head as compared to increase in discharge. There is nearly linear relationship between discharge and speed factor as observed in Fig.2.

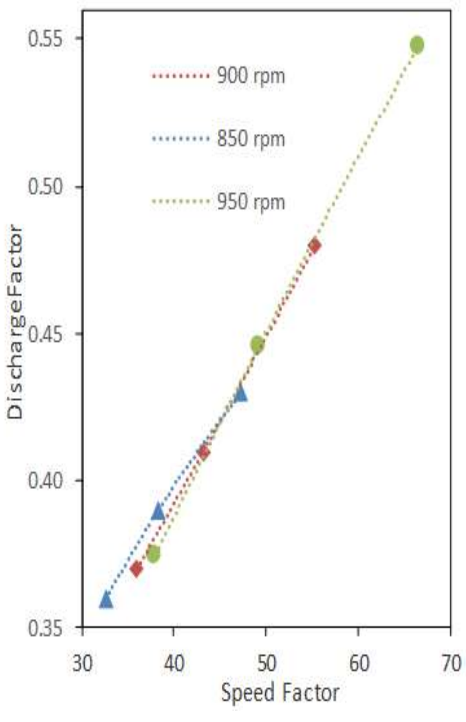

Fig.2 Variation of discharge factor

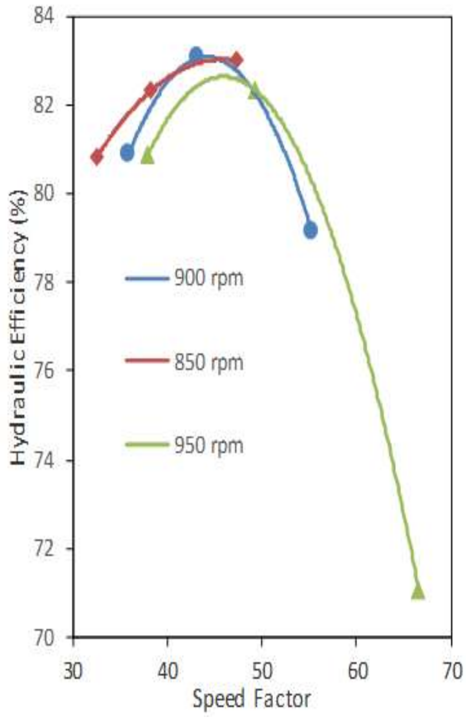

Fig.3 Variation of hydraulic efficiency

The variations of hydraulic efficiency with speed factor shown in Fig.3, is parabolic which resemble to the characteristic of an axial turbine. The turbine is found to have a maximum hydraulic efficiency of 83.05% at a rotational speed of 900 rpm for a discharge of 550 kg/s. With increase in rotational speed of runner, the efficiency curve shifts rightwards. The specific energy and degree of reaction decreases with increase in speed factor for all rotational as observed in Fig.4 and Fig.5, respectively. Further, specific energy and degree of reaction is more at lower speed and it is due to that more input energy is required to pass same discharge at lower speed. The degree of reaction at design operating conditions is found to be 0.75.

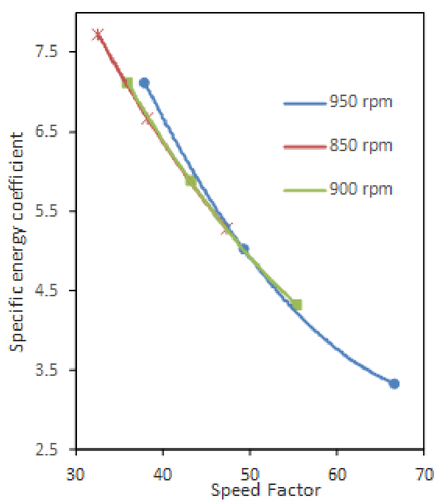

Fig.4 Variation of specific energy

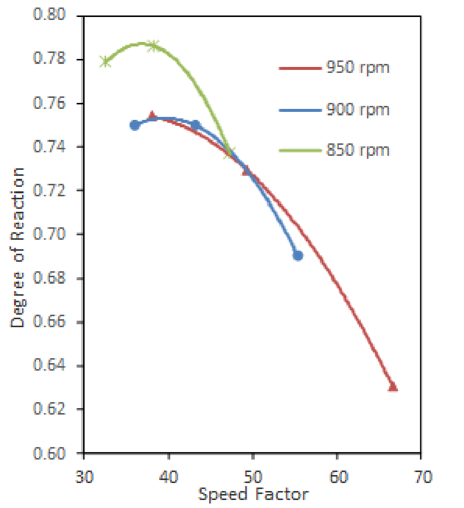

Fig.5 Variation of degree of reaction

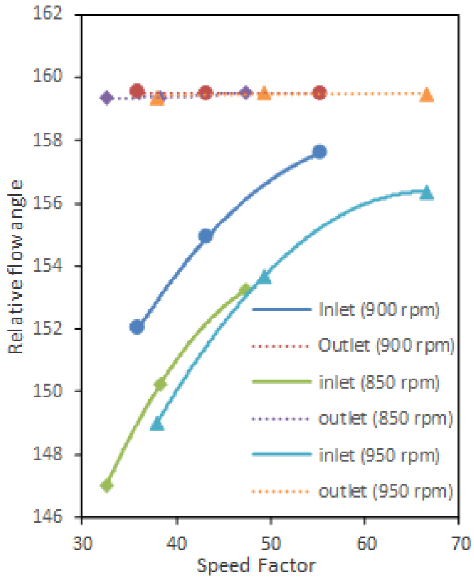

Fig.6 Variation of relative flow angle

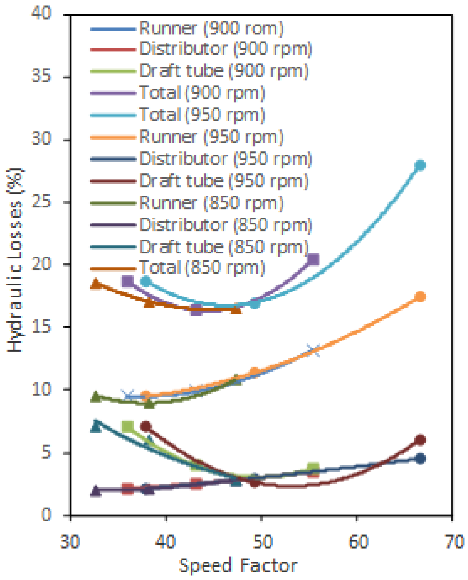

Fig.7 Variation of hydraulic losses

The variations of relative flow angles (with reference to tangential direction) at inlet and outlet of runner blade, shown in Fig.6, indicate that at inlet, the flow angle increases with increase in speed factor at all three speed. The highest value of flow angle is obtained at rated speed at different discharge and it is found to be less at either side of designed speed. At outlet, it remains independent of speed and discharge because the liquid leaves the blades tangentially. It is seen from Fig.7 that the loss in distributor and runner increases with speed factor i.e. decrease in discharge at all speeds but loss in draft tube has parabolic pattern with minimum value at design speed factor. Similarly, the curve representing total hydraulic losses is also parabolic with minimum value at design operating condition. It is also observed that maximum loss occurs in runner.

The summary of design and computed characteristic parameters for different operating conditions is given in Table 1.

Table 1 Variation of parameters at different operating conditions

Speed (rpm)	850			900			950		
Discharge (Kg/s)	500	550	600	500	550	600	500	550	600
η_h (%)	83.00	82.32	80.85	79.12	83.05	80.90	71.08	82.34	80.90
H	5.26	8.03	11.07	4.31	7.08	10.21	3.32	6.06	10.21
$H_{L\ Total}$ (%)	16.54	17.06	18.61	20.42	16.38	18.71	28.01	17.00	18.71
Ψ	5.28	6.67	7.72	4.33	5.88	7.12	3.34	5.03	7.12
SF	47.33	38.31	32.62	55.36	43.20	35.97	66.59	49.28	37.97
DF	0.43	0.39	0.36	0.48	0.41	0.37	0.55	0.45	0.37
R	0.74	0.79	0.78	0.69	0.75	0.75	0.63	0.73	0.75

The blade loading and blade to blade contours are given only for deign rotational speed. The pressure distributions i.e. blade loadings on runner blade shown in Fig.8 for different values of discharge at 900 rpm, indicate the greater load on blade towards leading edge which is in accordance design of axial flow runner. The blade loading increases with increase in discharge. The load on pressure side of blade decreases from leading edge to trailing edge while on suction side, pressure remains nearly constant except at leading edge. The peak in on suction side is seen due to mismatching of flow angles and it is least for design discharge. It is seen from the blade loading at three sections along the span of blade in Fig.9 that loading is nearly uniform along span but there is pressure peaks at mid and shroud indicating that there is mismatching of flow angles toward shroud. The uniform distribution of load from runner hub to its shroud is required in a hydraulic turbine for its stability otherwise there would be vibrations in the turbine leading to reduction in life of blades.

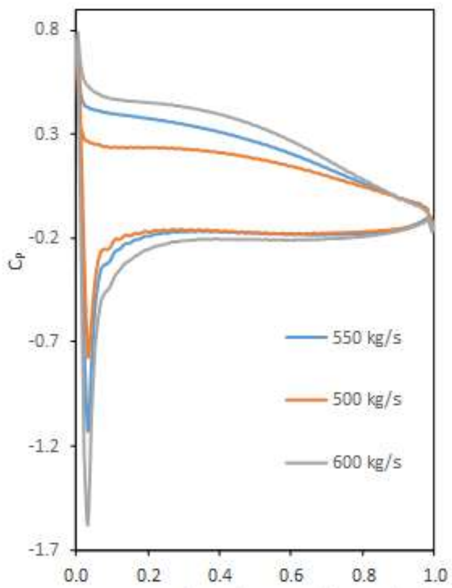

Fig.8 Blade loading at mid span for N = 900 rpm

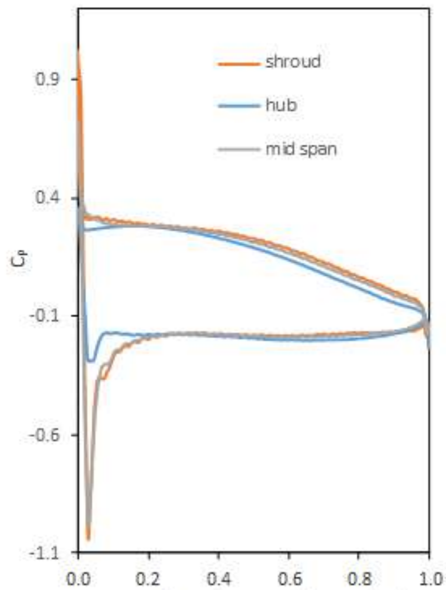

Fig.9 Blade loading at different sections along span

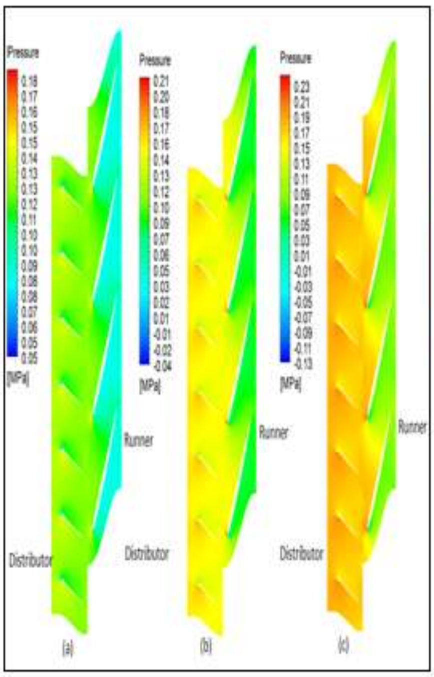

Fig.10 Pressure contours on blade to blade view at mid span at N = 900 rpm and at discharges of (a) 500 kg/s (b) 550 kg/s (c) 600 kg/

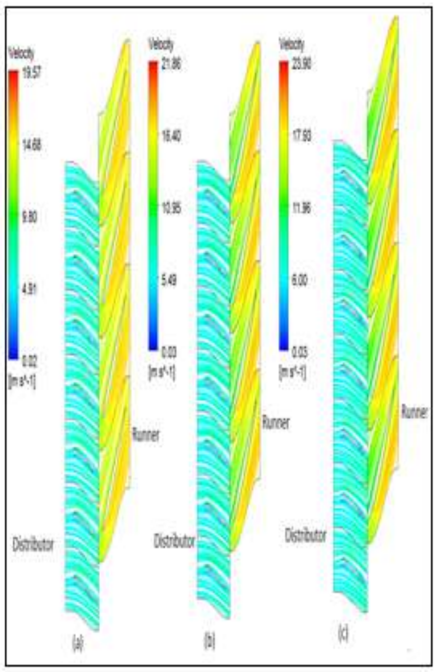

Fig.11 Streamline pattern in blade to blade view at mid span at N = 900 rpm and at discharges of (a) 500 kg/s (b) 550 kg/s (c) 600 kg/s

Pressure contours on blade to blade view at the mid span for design rotational speed and different discharge shown in Fig.10 indicate that pressure decreases from runner inlet to its outlet due to energy extraction. The pressure difference between two surfaces of runner blade increases with increase in discharge. The maximum value of pressure inside the runner increases with increase in discharge because of increased input energy. It is observed that pressure distribution is more uniform at design condition. The impact of water at leading edge is seen due to mismatching of flow angle and runner blade angle. It is observed from the streamline patterns on blade to blade view at design rotational speed and varying discharge, shown in Fig.11, that there is smooth entry to runner at design conditions. The velocity in runner increases with increase in discharge and it increases from inlet to outlet which opposite to that of pressure change.

VI. Conclusions

It is found from the design of Straflo turbine and its flow simulation that the performance characteristics obtained closely resemble to those of an axial flow turbine. The turbine is found to have maximum hydraulic efficiency of 83.05% for design operating conditions and it decreases at off design conditions. There is smooth flow to runner blade and the pressure distribution along span of runner blade is also uniform except at leading edge at design condition. The design can further be improved by modifying the geometries of runner blade and draft tube. The CFD is cost and time effective tool for efficient design and development of turbines.

References

[1]. I. Yuksel, Hydropower in Turkey for a clean and sustainable energy future, *Renewable and Sustainable Energy Reviews*, *12*(6), (2008), 1622-1640.

[2]. V. Prasad, Numerical simulation for flow characteristics of axial flow hydraulic turbine runner, *Energy Procedia, 14*(2012), 2060-2065.

[3]. S. Liu, J. Mai, J. Shao and Y. Wu, Pressure pulsation prediction by 3D turbulent unsteady flow simulation through whole flow passage of Kaplan turbine, *Engineering Computations, 26*(2009), 1006-1025.

[4]. Y. Wu, S. Liu, H. S. Dou, S. Wu and T. Chen, Numerical prediction and similarity study of pressure fluctuation in a prototype Kaplan turbine and the model turbine, *Computers & Fluids, 56*(2012), 128-142.

[5]. V.V. Barlit, P.Krishnamachar, M.M.Deshmukh, A.Swaroop and V.K.Gehlot, *Hydraulic turbines Volume 1 & 2*, (MANIT, Bhopal, 2007).