***International Journal of Research in Engineering and Science (IJRES)***
*ISSN (Online): 2320-9364, ISSN (Print): 2320-9356*
*www.ijres.org Volume 10 Issue 8 ǁ August 2022 ǁ PP. 265-273*

# Design and CFD Analysis of Francis Turbine

Prof Shivaji Kale[1], Shubham Kolase[2],
[1]*Prof Shivaji KaleDepartment of Mechanical Engineering, K.K.W.I.E.E.R*
[2]*Shubham KolaseDepartment of Mechanical Engineering, K.K.W.I.E.E.R*
[2]*KusumAhire Department of Mechanical Engineering, K.K.W.I.E.E.R*

***Abstract*** *-The aim of this design is to increase efficiency and avoid cavitation and fractures during the operation. In any hydroelectric power plant, hydraulic turbine plays a vital role which affects the overall performance of the plant and if utilized at suboptimal level, may lead to the loss of useful head. So, it becomes vital to predict the behaviour of hydro turbineunder actual working conditions. Experimental approach of predicting the performance of hydro turbine is costly and time consuming compared to CFD approach. Francis turbine has the efficiency of up to 90% but it is difficult to maintain this efficiency over a wide range due to various parameters such as material properties, vibrations, fluid flow, pressure fluctuations, area of contact, geometry of the turbine, stress at various locations where we may get versed with critical zones where the failure is supposed to occur. By the variation in some of these parameters it is possible to obtain an optimized design of the turbine. In Francis turbine maximum power is limited. When turbine operates at loads higher than design there is fluctuations, vibrations and power swings turbines are not supposed to operate in these conditions. To avoid this phenomenon operating range should be increased.*

***Keywords: - Francis turbine, CFD analysis, Vibrations, Cavitation***

Date of Submission: 01-08-2022 Date of acceptance: 15-08-2022

## I. INTRODUCTION

Hydropower, otherwise known as hydroelectric power, offers a number of advantages to the communities that they serve. Hydropower remains to play a crucial role in our fight against climate change by providing essential power, storage, and flexibility services. Hydropower provides low-cost electricity and durability over time compared to other sources of energy. Construction costs can even be minimized by using pre

existing structures such as bridges, tunnels, and dams. Some hydropower facilities can quickly go from zero power to maximum output. Because hydropower plants can generate power to the grid immediately, they provide essential backup power during major electricity outages or disruptions. Hydropower provides benefits beyond electricity generation by providing flood control, irrigation support, and clean drinking water. Hydropower is the largest, oldest and most efficient source of renewable energygeneration. Hydropower plants can be found in all over the world. Scientists are researching on developing and improving the power generation capacity of turbines. It is much needed for replacement of widely used fossil fuels. Hydropower plants turbines can have efficiencies up to 95% and maximum of 1.5km, depending on model type. Therefore, it is necessary to study the hydropower turbines for improving their performance.

The basis of the design of the turbine hydraulic passages is the velocity diagrams at the entry and exit of the turbine rotating element (called the runner). These lead to the Euler equation for theoretical torque and to the theoretical Euler efficiency of the turbine. Although elemental velocity triangles are employed for preliminary design of the hydraulic passages, for large turbines, model testing is necessary for verification of performance. Because of the cost and time involved in developmental model testing, more recently, a computerized finite element solution of the inviscid flow equations in the hydraulic passage, cross correlated with general data from model test results, is employed for advanced design. In particular, the efficiency of the hydraulic turbine must be optimized and established for contractual purposes.

## II. FRANCIS TURBINE

The Francis turbine is a mixed flow reaction turbine. This turbine is used for medium heads with medium discharge. Water enters the runner and flows towards the centre of the wheel in the radial direction and leaves parallel to the axis of the turbine.Turbines are subdivided into impulse and reaction machines. In the impulse turbines, the total head available is converted into the kinetic energy.In the reaction turbines, only some part of the available total head of the fluid is converted into kinetic energy so that the fluid entering the runner has pressure energy as well as kineticenergy. The pressure energy is then converted into kinetic energy in the runner.The electric generators which most often use this type of turbine have a power output which generally rangesjust a few kilowatts up to 800 MW.

### 2.1 Main Components of Francis Turbine:

- **Scroll Casing** (spiral casing):

In Francis Turbine, water from the Penstock enter a scroll casing that surrounds the runner.

- **Speed ring:**

From the scroll casing, water passes through a speed ring orstay ring which consists of a series of fixed vanes called stay vanes. The no. of stay vanes is usually taken as half the no. of guide vanes. The function of the speed ring is to direct the water from casing to the guide vanes or wicket gates and further resist the load imposed upon it.

- **Guide Vanes:**

From the speed ring, water passes through a series of guidevanes provided around the periphery (aerofoiled shaped) of the runner.Their function is to regulate the quantity supplied to the runner and direct the water on to the runner at an angle appropriate to the design.

- **Runner:**

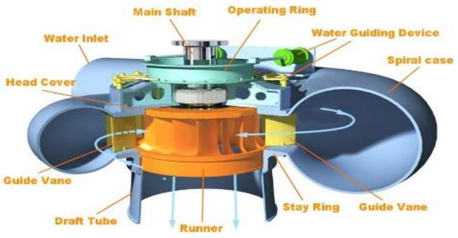
Francis Turbine blades are specially shaped that it has thin aerofoiled cross-sections when water flows over it, low pressure will be produced on one side and high pressure will be on another side. This will result in a lift force. Runner is a circular wheel on which a series of radial curved vanes are fixed. The surface of the vanes is made very smooth. The radial curved vans are so shaped that the water enters and leaves the runner without shocks. The runner may be cast in one piece or made of separate steel plates welded together. The runner made of CI for small output, cast steel, or stainless steel or bronze for large output. The runner blades should be carefully finished with a high degree of accuracy.

- **Draft Tube:**

A draft tube is a large pipe with an increasing cross-section area that connects the runner exit to the tailrace. The pressure at the exit of the runner of a reaction turbine is generally less than atmospheric pressure. The water at the exit cannot be directly discharged to the tailrace. A tube or pipe of the gradually increasing area is used for discharging water from the exit of the turbine to the tailrace. This tube of increasing area is called the draft tube.

- **Tailrace:**

The water after passing through the runner flows to the tailrace through a draft tube.



Francis Turbine

### 2.2 Efficiencies of Francis Turbines:

**1. Hydraulic efficiency:**

It is defined as the ratio of the power produced by the turbine runner and the power supplied by the water at the turbine inlet.

**2. Volumetric efficiency:**

It is possible some water flows out through the clearance between the runner and casing without passing through the runner. Volumetric efficiency is defined as the ratio between the volume of water flowing through the runner and the total volume of water supplied to the turbine.

**3. Mechanical efficiency:**

The power produced by the runner is always greater than the power available at the turbine shaft. This is due to mechanical losses at the bearings, windage losses and other frictional loss.

**4. Overall efficiency:**

This is the ratio of power output at the shaft and power input by the water at the turbine inlet.

### 2.3 Advantage of Francis Turbine:

1. The difference in the operating head can be extra simply controlled in Francis turbine than in the Pelton wheel turbine.
2. The ratio of utmost and least operating head can even be two in the case of Francis Turbine.
3. The mechanical efficiency of the Pelton wheel decreases faster by wear than Francis turbine.
4. Francis turbine variation in operating head can be more simply controlled.
5. No head failure occurs still at the low discharge of water.

### 2.4 Disadvantage of Francis Turbine:

1. The water which is not dirt-free can cause extremely rapid wear in high head Francis turbine.
2. As spiral casing is stranded; the runner is not simply available. Therefore, dismantle is hard.
3. The repair and inspection are much harder reasonably.

### 2.5 Applications of Francis Turbine:

1. Francis turbine is the most widely used turbine in hydro-power plants togenerate electricity.
2. Mixed flow turbine is also used in irrigation water pumping sets to pump water from ground for irrigation.
3. It is efficient over a wide range of water head and flow rate.
4. It is most efficient hydro-turbine we have till date.

## III. PROPOSED DESIGN

A Francis turbine is to be designed for the flow rate of 2 m3/s available at a project site at a net head of 10 m of water. The expected overall efficiency is 80%. The speed coefficient (or speed ratio) and the flow coefficient can be assumed as 0.8 and 0.6, respectively. The hydraulic losses in the turbine are 15% of available energy. Design the turbine rotor, with the salient dimensions and angles, to run at 300 rpm. The water leaves the rotor without any whirl component.

### 3.1 Calculations

$K_u= U_1/\sqrt{2GH}=0.8$
$K_f= V_{f1}/\sqrt{2GH}=0.6$
$V_{w2}=0$
Outer Periphery, Blade speed ($u_1$)
$U_1= (\pi D_1 N)/60$
$D_1=0.713mm$
As, $H_f= 0.6= V_{f1}/\sqrt{2GH}$
$V_{f1}= 8.4042m/s$
As, $Q= A_{f1}*V_{f1}$
$Q= \pi D_{1*}B_{1*} V_{f1}$
$B_1= 0.1062m$
By Velocity Triangle,
As, $\beta= 90°$ then $\sin ø = V_{f2}/V_{r2}$
$ø= 47.01°$
$\tan ø= V_{f1}/U_2$
$U_2= 7.3833m/s$
$U_2= (\pi D2N)/60$ then,
$D_2= 0.498m/s$

As, $Q= A_{f1}.A_{v1}= A_{f2}.A_{v2}$
Hence $B_2= (D_1.B_1)/D_2$
$B_2= 0.152m$
As, $\eta_h$= Runner Power/Hydraulic Power
$0.85= (V_{w1}.u_1 \pm V_{w2}.u_2)/g*H$
$V_{w1}= 7.441$ m/s
By Velocity Triangle,
$\tan\alpha= V_{f1}/V_{w1}$
$\alpha= 48.47°$
By Velocity Triangle, $\tan(180-\theta) = V_{f1}/(u-V_{w1})$
$\Theta= 114.12°$

***Table 1*** *Theoretical values*

| Sr. No. | Descriptions | Symbol | Values | Unit |
|---|---|---|---|---|
| 1 | Velocity of whirl at inlet | $V_{w1}$ | 7.441 | m/s |
| 2 | Velocity of whirl at outlet | $V_{w2}$ | 8.4042 | m/s |
| 3 | Tangential velocity of whirl at inlet | $U_1$ | 11.194 | m/s |
| 4 | Tangential velocity of whirl at outlet | $U_2$ | 7.3833 | m/s |
| 5 | Velocity of flow at inlet | $V_{f1}$ | 8.4042 | m/s |
| 6 | Velocity of flow at outlet | $V_{f2}$ | 8.4042 | m/s |
| 7 | Relative Velocity at Inlet of the runner | $V_{r1}$ | 11.489 | m/s |
| 8 | Relative Velocity at outlet of the runner | $V_{r2}$ | 11.489 | m/s |
| 9 | Vane angle at the exit. | Φ | 47.01° | |
| 10 | Vane angle at the inlet. | ɵ | 114.12° | |
| 11 | Guide vane angle | α | 48.47° | |
| 12 | Diameter Of Runner at inlet | $D_1$ | 0.713 | m |
| 13 | Diameter of Runner at outlet | $D_2$ | 0.498 | m |
| 14 | Blade thickness inlet | $B_1$ | 0.1062 | m |
| 15 | Blade thickness outlet | $B_2$ | 0.152 | m |

## IV. CFD ANALYSIS PROCEDURE

The process includes:

1. Formulate the Flow problem
2. Model the geometry and flow domain
3. Establish the Boundary conditions
4. Generate the Grid
5. Establish the Simulation Strategy
6. Establish the input parameters
7. Perform Simulation
8. Monitor the Simulation for Completion
9. Post-process the simulation to get the results
10. Make comparisons of the results
11. Repeat the process to examine sensitivities
12. Documents

### 4.1 Modelling of Francis turbine runner

We have designed the model of Francis turbine as per our calculated dimensions in CAD software in CATIAV5. First, we have drawn cross-section of blade as per calculated angle, thickness and height and in similar way we have drawn cross-sections at six planes and then we connected them by blend operation and making appropriate tangency and made blade profile.

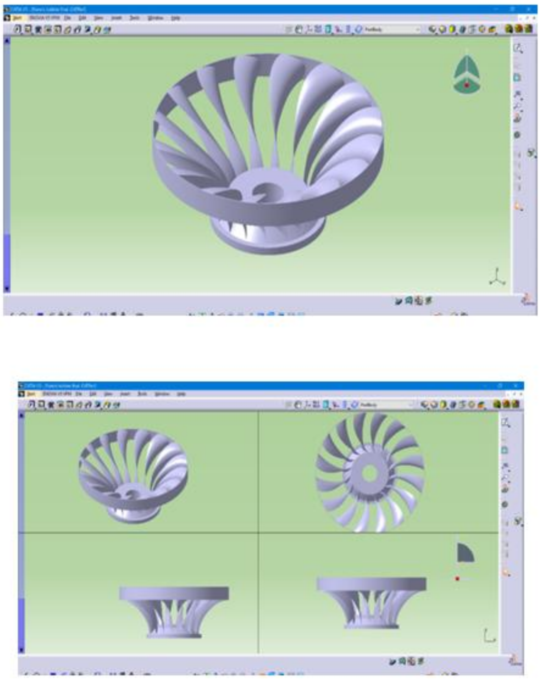


**Figure 1.** CAD model of runner

## V. CFD Analysis

Computational Fluid Dynamics (CFD) is the method of analysis by fluid flows using numerical solution methods. With the help of CFD, it is able to analyze complex problems involving fluid-fluid, fluid-solid or fluid-gas interaction. CFD has vast applications in engineering fields of aerodynamics and hydrodynamics, where quantities such as lift and drag or field properties as pressures and velocities are evaluated. Fluid dynamics is involved with physical laws in the form of partial differential equations. The process involves changing the physical laws into algebraical equations in order to efficiently solve the required equations numerically.

### 5.1 Phases of CFD analysis:

**1. Pre-processing**

Problem statement is transformed into an idealized and discretized computer model. Assumptions are made concerning the type of flow to be modelled (viscous/inviscid, compressible/incompressible, steady/non steady). Other processes involved are mesh generation and application of initial- and boundary conditions.

**2. Solving**

By performing actual computations in solving phase the computational power is obtained. There are multiple solvers available, varying in efficiency and capability of solving certain physical phenomena.

**3. Post-processing**

In the final phase, the obtained results are visualized and analyzed in the post processing phase. At this stage the analyst can verify the results and conclusions can be drawn based on the obtained results. Ways of presenting the obtained results are for example static or moving pictures, graphs or tables.

### 5.2 CFD ANALYSIS

We have done CFD analysis in SimScaleFluent. Firstly, we imported our model in Design modeler of SimScale. After importing to define path of fluid we made cylindrical enclosure around turbine. Such that fluid particles would flow through that enclosure. Enclosure is nothing but inlet, outlet and boundary for fluid flow. After being done with model, we have to mesh geometry. After meshing the elements generated are3616711. In CFD, we have to analyze pressure and velocity contouringas well as velocity vectors for turbine model. To do it, we have to use different Analysis System in SimScale Workbench which is Fluid Flow (Fluent).

First, start by meshing the model then apply the assumption and boundary conditions. For boundary condition, the model is placed in cylindrical enclosure simulation 10 million particles flowing from upper blade surface to bottom blade surface with varying velocity and pressure generating on blade surface. For boundary condition we named cylindrical enclosure faces as:

- Upper face- Inlet of flow
- Bottom face- Outlet of flow

- Cylinder surface- Wall
- The inlet boundary condition can be mass flow rate or velocity of fluid.
- The outlet boundary condition is pressure

### 5.3 Elements of Meshing:

Computational is divided into finite number of volumes or elements in meshing. Following important identities of meshing are mentioned.

- **Cell**: It controls volume into which domain is to be divide or broken into.
- **Node**: It is defined as a grid point.
- **Cell centre**: Centre of a cell.
- **Edge**: It is known as the boundary of a face.
- **Face**: Boundary of a cell. Face defines the cell zones (solid or fluid region) for two dimensional (2D) computational domains. Hence, CFD users has to define the fluid zone for the face. For, the 3D domain, the face can be a boundary like an inlet, outlet or wall etc.All essential faces should be named in meshing before exporting to the CFD solver.
- **Zone**:It is a grouping of nodes' faces and cells for, 3D domain, the volume region defines fluid or solid regions. All essential cell zones should be named either solid or fluid in meshing before exporting to CFD solver.
- **Domain**: It is known as the group of node, face and cell zones.

**Figure 2.** Meshing

*Table 2 Boundary conditions*

| Sr. No. | Parameters | CFD codes |
|---|---|---|
| 1 | Flow simulation domain | Single runner flow channel |
| 2 | Mesh | Structured |
| 3 | Fluid | Water at 25ºC |
| 4 | Inlet | Mass flow rate 2m$^3$/s |
| 5 | Outlet | Static pressure 0pa |
| 6 | Wall | No slip |
| 7 | Turbulence Model | k-Ω SST |
| 8 | Flow | Incompressible |
| 9 | Reference pressure | 1 atm |
| 10 | Water density | 996 kg/m$^3$ |

## VI. RESULTS OF ANALYSIS

Result has been calculated for flow rate of 2 m$^3$/s.

Results are as followed in screen shots of SimScale Fluent analysis workbench.

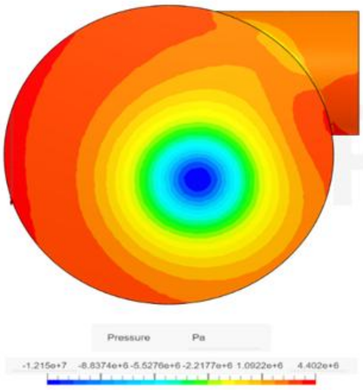


**Figure 3.** Pressure distribution

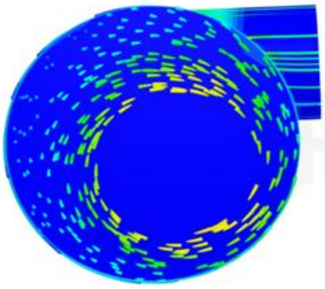


*Figure 4 Velocity Vector*

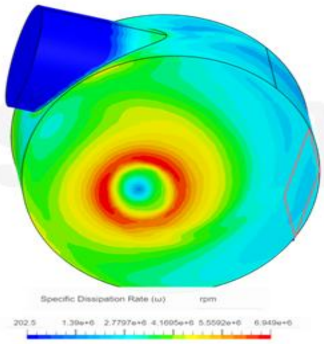


*Figure 5 Specific dissipation rate*

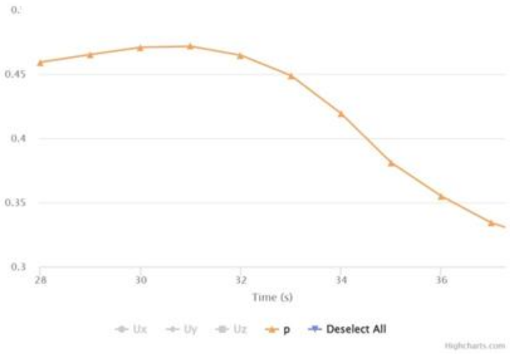


*Figure 6 Pressure Plot*

*Figure 7 Velocity Plot*

## VII. CONCLUSION

As the functioning of hydro projects depends greatly on hydraulic turbines, the design of these machines should be in sucha manner that it can sink with the electric grid demand efficientlyandeconomically. Maintaining higher efficiencies of hydraulicturbines with varying operating conditions according to the outputpower demands along with meeting economic viability isthe ultimate goal of the design cycle and performance analysis.The computational fluid dynamics (CFD) has emerged andbrought revolutionary changes in the field. With the help ofCFD, now it is possible to visualize the mechanisms of almostevery hydrodynamic phenomenon taking place during the turbineoperation which help in obtaining highly efficient designs. Boundary and initial conditions as well as turbulence modellingplay a pivotal role in obtainingaccurateCFD simulations of water turbines. Numerical prediction of phenomenon like cavitationrequires correct set of boundary and initial conditions along witha turbulence model. The experimental approach of evaluating the performance of Francis turbine is costly as well as time consuming. Conversely CFD approach is faster and large amount of results can be produced at virtually no added cost. The CFD approach for the prediction of efficiency of Francis turbine was developed with accomplishment of analysis of Francis Turbine performance. CFD analysis shows the distribution of various parameters like Pressure, velocity, stress at various points along the blade profile by using boundary conditions of pressure and mass flow rate at inlet and outlet. It can be concluded that CFD approach assists in reduction in cost of model testing and saving in time which leads to cost-effective analysis and may enhance the viability of hydropower development.

## VIII. FUTURE SCOPE

The scope of the upgrading the design of runner is to study the changes in velocity, pressure, stress induced by fluid flow in Francis turbine runner. The study or simulation is conducted of the hydraulic design of the new Francis type runner due to demanding request for unusually wide range of operation. The most challenging was the condition of fluid flow and geometry of runner blade, which was insufficient regarding specific speed of the new runner. The CFD computations confirmed that the presented design of the runner was successfully able to fulfil the expected parameters. The results of the optimized design were confirmed by simscale simulation with further studies of different parameters. The performance features of the new turbine were found as excellent especially applicable for wide range of operation heads and outputs. Even in the marginal operation there are not expected any troubles with the cavitation or excessive pressure pulsations and turbine vibrations.

## REFERENCE

[1]. Choi, G.W. K-Water 2014 Sustainability Report; K-Water: Deajeon, Korea, 2014.2.
[2]. Francis Turbine Blade Design on the Basis of PortArea and Loss Analy
[3]. Wu, J.; Shimmei, K.; Tani, K.; Niikura, K.; Sato, J. CFD-Based Design Optimization for Hydro Turbines.J. Fluids Eng. 2007, 129, 159–168.
[4]. Zhao Z. The cooling towers. Beijing, China: Water &Power Publishing House, 2001.
[5]. A highly efficient Francis turbine designed for energy recovery in cooling towers Daqing Zhou1, Huixiang Chen2 and Chunxia Yang2, 2015
[6]. Wei, Q.; Zhu, B.; Choi, Y.D. Internal Flow Characteristics in the Draft Tube of a Francis Turbine. J. Korean Soc. Mar. Eng. 2012, 36, 618–626. [CrossRef]
[7]. Shukla, M.K.; Jain, R. CFD Analysis of 3-D Flow for Francis Turbine. Int. J. Mech. Eng. 2011, 2, 93–100
[8]. Dr.Hermod Volume 13 Issue 5 Version 1.0 Year 2013 Nechleba, M. Hydraulic Turbine: Their Design and Equipment; Artia: Prague, Czechoslovakia, 1957.
[9]. Mei, Z.Y. Mechanical Design and Manufacturing of Hydraulic Machinery; Avebury Technical: Hants, UK, 1991.
[10]. Ingram, G. Basic Concepts in Turbomachinery; Bookboon: London, UK, 2009.

[11]. Gorla, R.S.R.; Khan, A.A. Turbomachinery: Design and Theory; Marcel Dekker: New York, NY, USA, 2003.
[12]. Wu, J.; Shimmei, K.; Tani, K.; Niikura, K.; Sato, J. CFD-Based Design Optimization for Hydro Turbines. J. Fluids Eng. 2007, 129, 159–168. [CrossRef]
[13]. Alnaga, A.; Kueny, J.L. Optimal Design of Hydraulic Turbine Distributor. WSEAS Trans. Fluid Mech. 2008, 2,175–185.