Turbine Test Rig to Investigate Flow Instabilities in Draft Tube

S Kumar1,*, S Khullar1, M J Cervantes2, and B K Gandhi1

1 Indian Institute of Technology, Roorkee, India
Department of Mechanical and Industrial Engineering, Indian Institute of Technology, Roorkee, Uttarakhand 247667
2 Lulea University of Technology, Sweden
Division of Fluid and Experimental Mechanics, Department of Engineering Sciences and Mathematics, Norrbotten 97187, Sweden
*Corresponding author: Tel.: +91 9417555636, mechengr03@gmail.com

Abstract: Time-dependent phenomena in a reaction turbine such as rotor-stator interactions (RSI) and rotating vortex rope (RVR) contribute to pressure fluctuations, vibrations, and even failure of the machine. A small scale test rig is being developed to investigate RSI and RVR and to study RVR mitigation methods. The test rig is designed for a maximum available head and discharge of 8 meters and 0.05 m$^3$/s, respectively. Provisions are made to operate the test rig in open and closed loops. The test rig is flexible to operate for the turbines of high specific speed (high head Kaplan turbine or very low head Francis) to low specific speed (high head Francis). The runner of the test rig is 200 mm in diameter at draft tube inlet. The turbine is connected with a variable speed generator to run up to 1000 rpm. This paper presents the design and arrangements of the test rig components according to the head and discharge conditions. Scale down design calculations for a model are performed using IEC 60193.

1. Introduction

The operational flexibility of Hydro Power Plants (HPP) is a necessity as it is required for stabilizing the grid due to their quick response to frequent changes in demand. Therefore, turbines especially reaction turbines and pump-turbine systems need to be operated over an extended operating range far from their Best Efficiency Point (BEP) [1]. At Part Load (PL) or High Load (HL) operating conditions, some flow instabilities occur in the draft tube which generates pressure pulsations in the turbine unit [1]. At off-design operation, two things are apparent: a) excessive swirl; and b) uneven distribution of velocity [3]. Excessive swirl may result in the strong coupling between the decelerating flow and axial pressure gradient, and flow starts separating at the draft tube center region which causes the formation of rotating vortex rope (RVR) [4]. The presence of RVR in the draft tube with a precessing frequency of 0.2-0.4 times of the runner, also called Rheingan’s frequency [5], and its harmonics cause high-frequency pressure pulsations, pulsative pressure recovery, power swings, etc., which lead to flow instabilities in the draft tube and reduce its performance. The decrease in the efficiency of a reaction turbine at the off-design condition is mainly related to the poor efficiency of the draft tube [6]. The other consequences include heavy vibrations and noise, which may cause high fatigue loading leading
to mechanical failure [7].

Since last century, extensive experimental measurements and numerical studies had been performed to investigate these instabilities in model reaction turbines by many researchers [2-9]. Active mitigation methods such as air or water injection in the draft tube were also proposed by many researchers [8-9]. A number of state of the art turbine test laboratories such as water power laboratory NTNU Norway, EPFL Switzerland, have been working continuously on these instabilities under projects like Turbine-99, Francis -99, FLINDT, FP7-Hyperbole [10-14] etc., which gave some useful information about the flow physics of RVR. However, the physical attributes of RVR such as vortex rope thickness, its eccentricity, and radius of formation from the center of the draft tube require further investigations. Moreover, the mitigation techniques need to be optimized for off-design (deep part load) operation to enhance the efficiency of the draft tube. A small test rig is better choice which will ease in measurement and operation to conduct the study on evolution and mitigation of vortex rope. The available model test rigs are required to maintain a number of operational parameters as per IEC 60193[15], and also need a large quantity of water for circulation.

This paper represents the hydraulic design of a new test rig that is being developed at IIT Roorkee, India to investigate the formation of the vortex rope at various PL conditions and its effects on the draft tube performance. The main purpose of the rig is to investigate the flow field-specific to conical diffuser part of the elbow draft tube and to evaluate the existing mitigation methods. These investigations will be conducted with the help of non-intrusive flow visualization techniques like Laser Doppler velocimetry or Particle Image Velocimetry and pressure measurements.

2. Selection of Turbine Unit

2.1. Selection of Physical Parameters

The designed test rig is flexible to accommodate both high as well as low specific speed turbines as per requirement. For the current study, a low head prototype Francis turbine operating at Nimoo Bazgo Hydropower plant, 3 x 15 MW, in Leh District of Jammu & Kashmir, India is taken as reference. The model is scaled down to 14.35:1 of the prototype exit runner diameter. The low head Francis turbine is chosen because it has lesser number of runner blades and is relatively easier to fabricate for such a small diameter as compared to a high head runner. The similarity laws according to IEC 60193 [15] were applied to fix the model parameters using dimensionless speed factor ($n_{ED}$), discharge factor ($Q_{ED}$), and power factor ($P_{ED}$). The relations used for calculation of these parameters are given below:

Speed factor:

$$n_{ED} = n * D/\sqrt{E}$$

(1)

Discharge factor:

$$Q_{ED} = Q/(D^2 * \sqrt{E})$$

(2)

Power factor:

$$P_{ED} = P/(D^2 * E^{3/2})$$

(3)

where $n$, $Q$ and $P$ are the rotational speed (rps), discharge (m$^3$/s) and power (kW) of the prototype, respectively. $D$ is the runner exit diameter and $E$ is the specific hydraulic energy of the prototype which depends on acceleration due to gravity (g) and rated head (H) as follow:

$$E = g * H$$

(4)

The runner exit diameter ($D_{in}$) and rotational runner frequency ($n_{in}$) of the model turbine were fixed as 0.2 m and 8.33 Hz, respectively. The calculations for other parameters as per equations. (1-4) are summarised below in Table 1:
| S. No | Parameters            | Prototype         | Model          | Dimensionless Factors |
|-------|-----------------------|-------------------|----------------|-----------------------|
| 1     | Rated Head            | 34.77 m           | 1.055 m        | Speed Factor \(n_{ED}\) | 0.517    |
|       | Exit runner diameter  | 2.87 m            | 0.2 m (fixed)  | Discharge Factor \(Q_{ED}\) | 0.308    |
| 2     | Power                 | 15000 kW          | 0.385 kW       | Power Factor \(P_{ED}\) | 0.289    |
| 3     | Rotational speed (rps)| 3.33              | 8.33 (fixed)   | Speed number \(\Omega\) | 1.075    |
| 4     | Discharge             | 47 m\(^3\)/s     | 0.03976 m\(^3\)/s |                |          |

From Table 1, the discharge at BEP is 0.03976 m\(^3\)/s by applying the similarity laws. This was further verified theoretically as per the guidelines given in Hermod and Brekke [16]. The following methodology was adopted to calculate the theoretical discharge at BEP.

The gross head of the turbine is 2.1 m after considering the losses in the penstock. The average velocity \(V_a\) at the inlet of the spiral casing is calculated using eq. (5).

\[
V_a = u_c \times \sqrt{\frac{H_g}{2g}}
\]

where for steel spiral casing \(0.7 < u_c < 0.8\), \(H_g\) is the gross head.

The inlet diameter of the scaled-down spiral case is 248 mm and the designed flow rate at the inlet of the spiral case is calculated as 0.049 m\(^3\)/s, as per eq. (6).

\[
Q = \frac{\pi}{4} \times D^2 \times V_a
\]

As per the guidelines [17], this design discharge is 1.2 times of BEP discharge \(Q_{BEP}\) (eq. 7).

\[
1.2 \times Q_{BEP} = \text{Design discharge}
\]

Thus, the theoretical value of discharge at BEP comes around 0.04083 m\(^3\)/s, which is close to the discharge obtained by similarity laws.

To avoid cavitation at the runner outlet, the setting height of the turbine is determined with respect to head and tailwater levels. Net Positive Suction Head (NPSH) is estimated at BEP using eq. (8).

\[
\text{NPSH}_{\text{required}} = a \frac{C_m^2}{2g} + b \frac{U_2^2}{2g}
\]

where \(a\) and \(b\) are coefficients that depend upon the speed number. For the above turbine model, as per guidelines in [16], the value of “a”, and “b” are taken as 1.12 and 0.1, respectively. \(C_m\) is the axial velocity of the model turbine, and \(U_2\) is the peripheral velocity of the runner at the exit. For cavitation free operation, the maximum value of the setting height should not be more than 9.82 m, as calculated by eq. (9).

\[
H_s < H_{\text{atm}} - H_v - \text{NPSH}_{\text{required}}
\]

2.2. Preliminary CFD Simulations

The 2D drawings of the model were provided by Bharat Heavy Electricals Limited (BHEL), India, which is a renowned manufacturer of hydro turbines in India. After scaling down the whole geometry, a 3D CAD model of the turbine components was made for the preliminary computational fluid dynamics (CFD) simulations to get the assurance that the vortex rope was generated in the draft tube for such a small head and exit runner diameter. These simulations were conducted by generating a tetrahedral mesh of all the components of the turbine unit in ANSYS ICEM CFD to expedite the
process. The orthogonal quality of the mesh was kept above 0.3 and the whole domain was discretized into 17.16 million elements. The mesh of the model components is shown in Fig.1 (a-d). The transient simulation was performed using ANSYS CFX. The flow rate and pressure were defined at inlet and outlet boundaries, respectively. The SST $k$-$\omega$ model was used to model turbulence quantities as it is most appropriate for flows with high adverse pressure gradients compared to the other two-equation RANS models [18]. The results of the simulation are shown in Fig. 2(a-b). The iso-surface in Fig. 2(a) shows that the vortex rope is formed at 60% PL operating condition and Fig. 2(b) shows the stagnation and backflow velocity regions in the draft tube.

3. Hydraulic Design

After fixing the turbine physical parameters, the hydraulic design of other test rig components, namely lower tank (sump), upper tank and piping system were carried out. The discharge at BEP is 0.03976 m$^3$/s which is being pumped from the lower tank to the upper tank by 2 pumps. The discharge at BEP is divided equally in each pump. According to the pumping station design [19], the recommended velocity in the delivery pipe ranges from $0.6 \leq V_a \leq 2.7$ m/s for the flow rate of less than 0.315 m$^3$/s.

The desired velocity was chosen as 1.7 m/s which is an average of the above-mentioned range. The diameter of the pipe for this velocity and discharge at BEP was calculated by applying the continuity eq. (10) which gave 172.565 mm.

$$Q_{pipe} = \frac{\pi D_{pipe}^2}{4} \times V_{pipe}$$  \hspace{1cm} (10)

The discharge at BEP is pumped through pipes by 2 pumps and the equivalent hydraulic diameter of each pipe comes around 122 mm. The nearest standard pipe size available is 100 mm, which was chosen for each pump in the test rig. The velocity in a 100 mm diameter is 2.53 m/s, which is in acceptable range as per design guidelines [19]. Water pumped from both pipes is merged into a 150 mm pipe which again diverges in two separate 100 mm pipes at the inlet of the upper tank.

Figure 1. (a) Spiral casing with stay vanes; (b) Draft tube; (c) Runner with hub; (d) Guide vanes; (e) Pressure iso-surface contour; (f) Velocity contours
Small storage of water is provided through an upper tank (Fig. 3a) which acts like a pondage or forebay of runoff river Hydro Power Plant (HPP) and has a capacity of approximately 10 seconds storage with 0.407 m$^3$ of water volume. The upper tank is 800 mm in diameter and 1 m high. The inlet pipes are coming through the bottom of the tank because it minimizes the fluctuations in water. A screen is placed in the upper tank to provide a constant head and the excess water overflows back to the sump. To operate the rig in an open loop, the screen of the upper tank has a height of 900 mm, and a gap of 100 mm is provided at the top to maintain a constant water level. A bell mouth with an elliptical profile [20] is used in the center of the upper tank to guide the water towards penstock which has a diameter of 150 mm.

The submergence depth of the bell mouth in the tank is 700 mm below the water level surface, which is determined by eq. (11) [21]
\[
h = 0.5 + 2 \times F_r \frac{D}{D}
\]
where, \( F_r = \frac{V}{\sqrt{gD}} \), and D is the diameter of the pipe connected to the bell mouth.

The depth of submergence comes out as 691 mm from eq. (11) to avoid air entrainment at the intake of the bell mouth. To validate the calculations, numerical simulation was performed for three different depths: 600 mm, 650 mm, and 700 mm from the top surface of the water level, created with the help of a screen in the upper tank, as shown in Fig. 3(b-d). The numerical simulations were performed using ANSYS CFX at mass flow equals to 1.1 times of BEP mass flow. The tank was first filled for 10 seconds to achieve the desired height of the water level and then the outlet from the turbine pipe is opened. The results showed that at a depth of 650 mm, the air entrainment is clearly visible with entrained air capsules in the bell mouth and turbine inlet pipe flow, but at 700 mm and 650 mm depths, negligible or no air entrainment is observed. The traces of dissolved air present for these two cases can be removed by a vacuum pump operating around ± 0.5 bar pressure present in the lower tank during steady operation. The present physical dimensions of the tank suggest that it is able to damp the turbulence entering the tank due to pumping as the water level at the top has negligible fluctuations. The lower tank acts as a sump for the test rig and has a volume of around 4 m$^3$ which is necessary to decrease the draft tube turbulence, and maintain the rise in water temperature (not more than 2$^\circ$ to 3$^\circ$ C in about 2 to 3 hours continuous operation in loop) due to hydraulic losses. The tank is divided in two sections by a screen whose length is two-third of the tank total length. An arrangement of the chilling unit is also incorporated in the lower tank to maintain the water temperature as desired. Numerical simulation is carried out for the lower tank to observe the flow velocity, turbulence and water level fluctuation at the same mass flow rate as considered for upper tank simulation. The velocity contours at the midplane of the lower tank are shown in Fig. 4. The turbulence of the incoming flow is dampened by the perforated semicircular screen. The water in the lower tank is steady and free from turbulence when it is pumped from the lower side of the tank.
Figure 3. (a) Upper tank layout; Air volume fraction contours in the mid plane for bellmouth at depth of (b) 700 mm; (c) 650 mm; and (d) 600 mm

The selection of pumps requires the estimation of hydraulic losses in the test rig and determination of the operating range from the functional characteristic curve which is shown in Fig. 5. Two vertical centrifugal pumps with integrated variable speed drive (GRUNDFOS make, flow rate 0-0.03 m³/s each) were chosen for the test rig water circulation with the arrangement of series and parallel operations. The material for the test rig is selected as stainless steel to avoid corrosion etc.

Figure 4. Velocity contours in lower tank

Figure 5. Functional characteristic curve of pump
Figure 6. Side view of the test rig.

The complete schematic layout in the wireframe is shown in Fig. 6. In this test rig, three expansion joints are mounted for a minor adjustment in the length of the pipes. A provision for a bypass line is also provided for closed-loop operation of the test rig.

An electromagnetic flowmeter, torque sensor, pressure transmitters for head measurement, piezoresistive pressure sensors in the draft tube along with PIV will be installed for the measurements to investigate the RVR phenomena. All the instruments will be connected through a PXI system for real-time data logging. A pressure equalizer will be mounted at penstock and another at the exit of the draft tube. The connections and arrangement of the pressure equalizer are according to IEC60041 [17]. A submersible pressure transmitter will be installed in the upper tank to measure the water level, while in the lower tank an Ultra Sonic Level sensor (ULS) will be installed.

4. Major highlights of the turbine test rig
Both theoretical hydraulic design and numerical studies of the test rig components complement each other and enable the experimental setup to achieve its objectives.

- Arrangement for in-situ instrument calibration.
- Provisions for passive and active mitigation mechanisms.
- The small size of the test rig will ease the non-intrusive measurements along with other instruments.
- The design of the upper tank and lower tank arrests the turbulence and fluctuations which are generated by the two pumps.
- Real-time data logging and computer control systems will also guarantee a highly accurate measurement.

Acknowledgment
The authors are thankful to financial support by the Central Power Research Institute (CPRI), Government of India, through a National Perspective Plan (NPP) grant. The authors also acknowledge technical support provided by BHEL (our industrial partner in project) along with the model drawings for investigation on the flow instabilities in the draft tube.

References
[1] Trivedi C, Cervantes M J, Gandhi B K and Dahlhaug G O 2014 Journal of Hydrodynamics, Ser. B 26(2) 277-290.
[2] Goyal R, Cervantes M J and Gandhi B K 2017 Journal of Fluids Engineering 139(4) 041102.
[3] Falvey H T, 1971, US Bureau of Reclamation Report, REC-ERC-71-42
[4] Faler J H and Leibovich S 1977 The Physics of Fluids 20(9) 1385-1400.
[5] Rheingans W J 1940 Trans. ASME, 62(3) 171-184.
[6] Dörfler P, Sick M and Coutu A 2012 *Flow-induced pulsation and vibration in hydroelectric machinery: engineer’s guidebook for planning, design and troubleshooting* Springer Science & Business Media.

[7] Casanova G and Mantilla V C A 2010 *Revista Facultad de Ingeniería Universidad de Antioquia, 55* 90-98.

[8] Muntean S, Susan-Resiga R, F Câmpian, V C, Dumbravă C and Cuzmoș A 2014 *UPB Sci. Bull., Series D 76(3).*

[9] Bosioc A I, Susan-Resiga, R, Muntean S and Tanasa C 2012 *Journal of Fluids Engineering, 134(8)* 081104.

[10] Andersson U and Cervantes M 2008 *In IAHR Symposium on Hydraulic Machinery and Systems.*

[11] Trivedi C, Gogstad PJ and Dahlhaug OG 2017 *Journal of Renewable and Sustainable Energy 9(6)* 064502.

[12] Iliescu MS, Ciocan GD and Avellan F 2008 *Journal of Fluids Engineering* 130(2).

[13] Favrel A, Liu Z, Takahashi W, Irie T, Kubo M and Miyagawa K 2019 *In IOP Conference Series: Earth and Environmental Science 405(1)* 012035.

[14] Favrel A, Gomes Pereira Junior J, Landry C, Müller A, Nicolet C and Avellan F 2018 *Journal of Hydraulic Research 56(3)* 367-79.

[15] IEC, 1999, *International Electrotechnical Commission, Geneva, Switzerland, Standard No. IEC, 60193.*

[16] Brekke, H 2000 NTNU, Trondheim.

[17] IEC 60041 1991 *International Electrotechnical Commission, Geneva, Switzerland, Standard No. IEC, 60041 ed.3.*

[18] Jost D, Skerlavaj A and Lipej A 2012 *IOP Conference Series: Earth and Environmental Science 15,* 062016.

[19] Jones G M, Sanks R L, Bosserman B E and Tchobanoglous G 2006 *Pumping station design* Gulf Professional Publishing.

[20] Blair G P and Cahoon W M 2006 *Race Engine Technology* 17 34-41.

[21] Indian Standard Codes IS 9761 1995 [WRD 14: water conduct system]