Francis Turbine Analysis Between Computational Fluid Dynamics (CFD) and Experimental Methods

Muhammad Naim bin Md. Kamal, Khairul Shahril bin Shaffee, Mohamad Sabri bin Mohamad Sidik, Ahmad Razlee Ab. Kadir and Johan Ihsan bin Mahmood

Abstract Hydroelectric power has become the most promising source in the power sector to sustain the growth of any nation. In any hydroelectric power plant, the 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 behavior of the hydro-turbine under actual working conditions. Francis turbines are the most well-known water turbines being used today. The Francis turbines works in water depths from 10 to 650 m (33–2133 ft) and are fundamentally utilized for electric power generation. This research consists of a simulation process and experimental research in order to compare both of the results. The geometry is modelled using the CATIA software and transferred into Ansys for the analysis. All the main parts that are included in the Francis turbine educational kit at the Universiti Kuala Lumpur Malaysian Spanish Institute such as the spiral casing, the runner blade, guide vane and the draft tube is constructed in the 3D model. The highest accuracy for the Francis turbine occurs at 1300 RPM and the highest inaccuracy percentage is within 30% and the lowest inaccuracy percentage is within 2%.

Keywords Francis turbine · CFD · CATIA · Ansys

M. N. bin Md. Kamal · K. S. bin Shaffee (✉) · M. S. bin Mohamad Sidik
A. R. Ab. Kadir · J. I. bin Mahmood
Malaysian Spanish Institute, Universiti Kuala Lumpur, Kulim Hi-Tech Park, 09000 Kulim, Kedah, Malaysia
e-mail: khairuls@unikl.edu.my

M. S. bin Mohamad Sidik
e-mail: msabri@unikl.edu.my

A. R. Ab. Kadir
e-mail: ahmadrazlee@unikl.edu.my

J. I. bin Mahmood
e-mail: johanihsan@unikl.edu.my

A. Ismail et al. (eds.), *Advanced Engineering for Processes and Technologies*, Advanced Structured Materials 102, https://doi.org/10.1007/978-3-030-05621-6_14

1 Introduction

Energy plays an important role in providing a steady growth of a country as well for the economy growth as the increase of citizen is also resulting in a higher power demand. As in Malaysia, the requirement of hydroelectric dam as power generators is increasing because of the suitable geographical terrain in Malaysia. The demand will continue to increase as the population of Malaysia continues to increase. The data of hydroelectric dam capacity in Malaysia from 2004 shows that hydro power generation consists 11.0% in the Peninsular Malaysia [1].

The Francis turbine educational kit simulates the condition of a real Francis turbine that is used in power generation in a real life condition. It is able to give almost the real experience in the understanding how the Francis turbine operates. By running the experiment, data from the Francis turbine educational kit and with some theoretical calculations from the data, the efficiency of the turbine can be obtained. The aim of this study is to compare efficiency data result from the conducted experiments using the Francis turbine educational kit and from running simulations using the Ansys computational fluid dynamics (CFD) code and to validate the data that were obtained in a real life condition and simulations condition.

Even though it is possible to predict turbine characteristics by model tests in the laboratory, time and budget limitations, prototype restrictions promoted the use of CFD tools for the turbine optimization. Developing technology enhanced the computational power and led to an improvement in the turbine design. An accurate prediction of the flow inside the hydraulic turbine is nowadays possible by the use of state-of-the-art CFD tools [2]. The latest CFD tools are the outcome of several researches handled during the last four decades. Validation of the lately developed tools proves that the accuracy of CFD tools are very high. This made the power of CFD tools undeniable in the design process [3].

One of the most desirable types of water turbine that is in use in power production is the Francis turbine. The Francis turbine is a type of water turbine that was developed by James B. Francis in Lowell, Massachusetts [4]. It is an inward-flow reaction turbine that combines the radial and axial flow concepts. Figure 1 shows the design of a Francis water turbine and the main components in the Francis turbine.

Francis turbines are the most well-known water turbines being used today. The Francis turbine works in water depths from 10 to 650 m (33 to 2133 ft) and is fundamentally utilized for electric power generation. The turbine controlled generator force yield large runs from 10 to 750 MW, however small hydro establishments may be lower. Penstock (input pipe) widths are between 1 and 10 m (3 and 33 ft). The speed rate of the turbine is from 83 to 1000 rpm [5]. The entryways around the outside of the turbine's pivoting runner conform the water stream rate through the turbine for diverse water stream rates and force generation rates. Francis turbines are just about constantly mounted with the pole vertical to keep water far

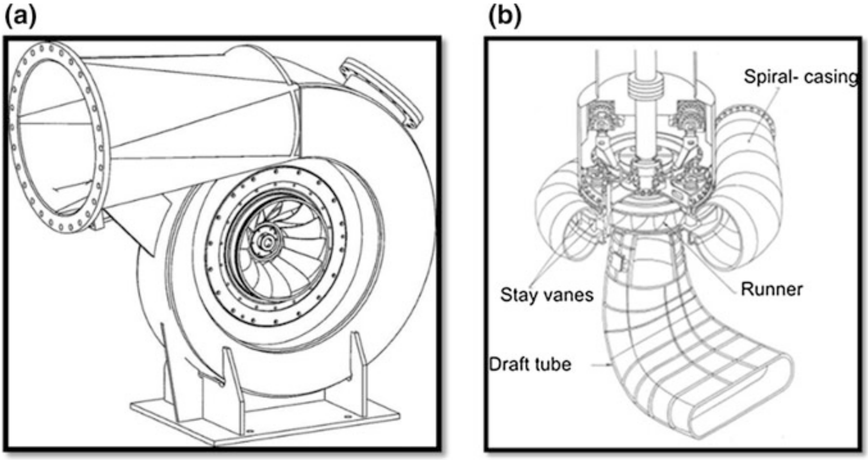

Fig. 1 **a** Francis turbine; **b** Francis turbine main component

from the connected generator and to allow installation and maintenance access to the turbine.

The Francis turbine is a kind of response turbine, a class of turbine in which the working liquid goes to the turbine under huge pressure and the energy is transferred to the turbine blade from the working liquid. A part of the energy is transferred from the fluid because of the pressure change in the blade of the turbine, quantified by the outflow of degree of reaction, while the remaining quantity of the energy is concentrated by the volute casing of the turbine. At the exit, the water follows up on spinning cup-shape runner features, leaving at low speed and low swirl with very little kinetic and potential energy left. The turbine's draft tube is molded to help decelerate the water stream and regain the pressure.

The centrifugal pump data were observed at a monitor that has been provided from the manufacturer of the Francis turbine educational kit. The following formula shows a theoretical calculation how the data is displayed on the monitor.

$$Output\,Power\;(\mathrm{W}) = \rho g h Q = PQ \tag{1}$$

$$Input\,Power\;(\mathrm{W}) = Torque\;(\mathrm{Nm}) * Average\,speed\;(\mathrm{rad/s}) \tag{2}$$

$$Average\,speed\;(\mathrm{rad/s}) = Average\,speed\;(\mathrm{rpm}) * 2\pi/60 \tag{3}$$

$$Efficiency = Output\,Power/Input\,Power \tag{4}$$

Turbine Equation

$$Load\ (\mathrm{N}) = F1 - F2 \tag{5}$$

$$Average\ speed\ (\mathrm{rad/s}) = Average\ speed\ (\mathrm{rpm}) * 2\pi/60 \tag{6}$$

$$Torque\ (\mathrm{Nm}) = Load * radius\ of\ the\ wheel\ (0.026\ \mathrm{m}) \tag{7}$$

$$Input\ Power\ (\mathrm{W}) = \rho ghQ = PQ \tag{8}$$

$$Output\ Power\ (\mathrm{W}) = Torque\ (\mathrm{Nm}) * Average\ speed\ (\mathrm{rad/s}) \tag{9}$$

$$Efficiency = Output\ Power/Input\ Power \tag{10}$$

Note:

Pressure, P = Reading from pressure gauge (must convert from bar to Pa)
Flow rate, Q = Must change from LPM to m^3/s

CFD has emerged out as a powerful tool for predicting the performance of mechanical bodies subjected to dynamic flow conditions [6]. There are ample of evidences where analysts at various levels have taken the advantage of this tool to solve so many problems related to performance analysis. There is a lot of research that has been conducted in simulation analysis of the Francis Turbine using the CFD method such as the research done by ČARIJA where the research was conducted for a dam that is located near Rječina by constructing a 3D model for a simulation in Ansys CFD to find the efficiency data for optimization of the dam. The result that was obtain from the research is that the inaccuracy of computed results for all analyzed characteristic values was within ±2% over the whole analyzed operating range, increasing in accuracy towards the turbine optimal efficiency point to less than 0.5% inaccuracy [7]. As for a research that was conducted by Akin was to research a methodology for conducting an analysis using CFD for a

Table 1 Data of boundary condition for simulation

Boundary Condition	Value in experiment	Value in Ansys
Inlet	The inlet for Francis turbine	Define the inlet location in Ansys
outlet	The outlet for the Francis turbine	Define the outlet location in Ansys
wall	The runner blade of the Francis turbine	Define the runner location in Ansys
Flow rate	125.6 (Lpm)	5 (m/s)
Material	water	water
Guide vane opening	25% opening	25% opening
Average speed (rpm) of the runner blade	2300–700	Same with experiment data but unit is in rad/s

Francis turbine where the result for the developing design methodology was applied for the turbine design of an actual hydropower project. The overall results of each Francis-type turbine components designed with the help of CFD are presented in Table 1. According to the final CFD results, an overall turbine efficiency of 92.3% is reached. As the performance values of the design satisfy the requirements of the Yuvacik H.E.P.P project, the structural verification of the design is accomplished and the manufacturing process was started [5].

2 Methodology

This research consists of a simulation part and an experimental research in order to compare both of the result. Figure 2 shows the flowchart for planning the research from the start until the end of this research.

2.1 Geometry Modelling/Boundary Condition

2.1.1 Geometry Modelling

In order for the simulation to be successful, a geometry modelling is required with a scale that is the same as the Francis turbine educational kit. The geometry is modelled using the designing software CATIA before the model can be transferred into Ansys for further simulation. All the main parts that are included in the Francis turbine educational kit such as the spiral casing, the runner blade, guide vane and the draft tube is constructed in the 3D model so that the model can be transferred into Ansys software to perform the simulation. Figure 1 shows the 3D design, and then transferred model from CATIA into Abaqus designing software to do some

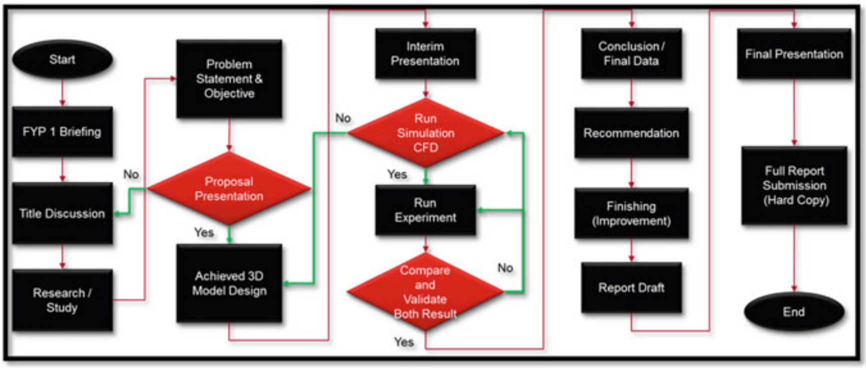

Fig. 2 Flowchart of research process

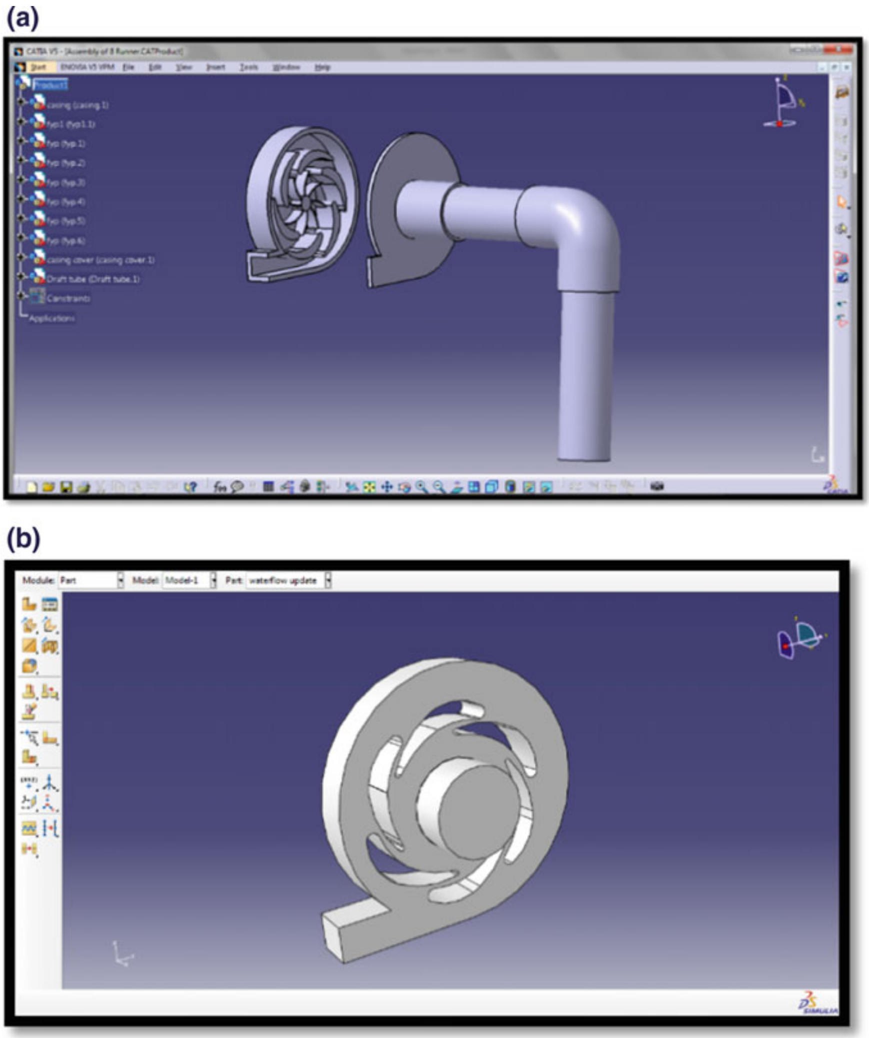

Fig. 3 **a** Complete assembly of Francis turbine in CATIA; **b** Final design of 3D model

Booleans operations to so that the final design can act as containment of the water for the simulations in Ansys to be successful. Ansys CFD can only read a fully closed system to be run. Figure 3 shows the final 3D design of the Francis turbine educational kit and the main parts that has been constructed into 3D model.

2.1.2 Boundary Condition

The final design is transferred into the Ansys software to run the simulation using Fluid Flow (Fluent). In Ansys the parametric data that need to be defined such as the inlet, outlet, and wall for the blade so that the data after the simulations can be interpreted. The boundary conditions for the simulation to be run will be set the same as the experimental setup of the Francis turbine educational kit, i.e. input values such as the flow rate, average speed (rotation per minute), guide vane opening, and material for the simulation to run which is water. Table 1 shows the parametric data and the boundary conditions that were used during the simulations in the Ansys CFD software.

2.2 Experimental Study

The experiments were executed based on the procedure that was stated in the lab manual of the Fluid lab. The procedure of this experiment and Fig. 4 of the Francis turbine educational kit is shown below.

2.2.1 Procedure

The apparatus was set up with the Francis turbine on the bench top and the computer as stated in the apparatus setup. The control valve opened and slowly closed the bypass valve. The adjustable handle guide vane was adjusted from the Francis

Fig. 4 Francis turbine educational kit

turbine to be half opened. The water was directed by the guide vane wall and caused the turbine to rotate. Then, the frequency was switched to 50 Hz by turning the turning knob of the frequency motor. The system was allowed to run for about 3 min to let the flow rate reading become stable. The last procedure was to repeat the same step but different types of guide vane opening in order to determine which type of opening will give the highest efficiency value. The data from the experiment was captured by the software in the computer. The procedure on how to use the software in the computer is shown below.

Turn on the computer for the first step and next is to start the program by double click the software icon. Next is to select the Francis turbine experiment icon. Click on the start button icon that is located at the bottom left of the program window. After finishing the experiment click on save file to save the file in the desired location. Lastly ensure that readings are available to run the experiment.

3 Results and Discussion

3.1 Simulations Result

There are several results that have been obtained during the simulations process which is depending on the guide vane opening. These results are selected based on which guide vane opening caused the highest efficiency in this case the guide vane opening is 25%. The results that obtain during the simulations process are shown in the Table 2.

Based on the result there are some consistencies from the data that has been obtained and the Fig. 5 shows the data of the average speed (rpm) versus the efficiency (%).

Table 2 Data of simulation

Average speed (rpm)	Average speed (rad/s)	Torque (Nm)	Output power (W)	Input power (W)	Efficiency (%)
1498	156.87	0.11617676	18.22467078	102.8389	0.17721573
1397	146.29	0.11702528	17.12003755	100.7401	0.16994263
1301	136.24	0.1172991	15.98087647	96.5426	0.16553186
1207	126.40	0.11834923	14.95895738	94.4439	0.15838987
1107	115.92	0.11864739	13.75417127	94.4439	0.14563324
1000	104.72	0.11882454	12.44327673	90.2464	0.13788114
901	94.35	0.1190762	11.23513708	88.1476	0.12745823
807	84.51	0.1194553	10.09502912	86.0489	0.11731735
706	73.93	0.11943101	8.829791001	81.8514	0.10787587

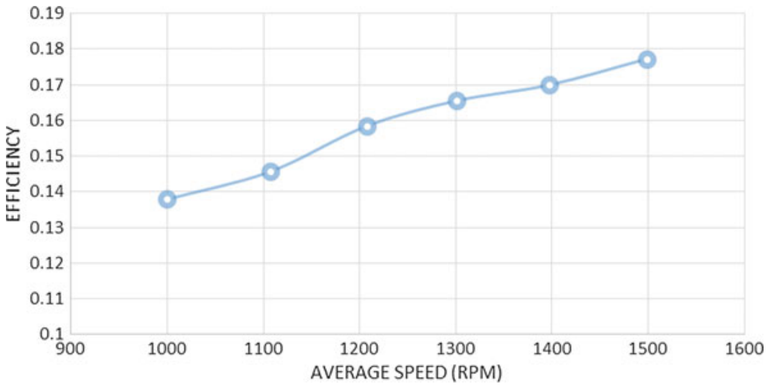

Fig. 5 Rpm versus efficiency (simulation)

From Fig. 5, the data of simulation is directly proportional between the efficiency and the average speed (rpm). As the average speed (rpm) was increasing so was the efficiency.

3.2 *Experiment Results*

There are several results that were been obtained during the experimental process which is depending on the guide vane opening. These results are selected based on which guide vane opening caused the highest efficiency in this case the guide vane opening is 25%. The result that obtained during the experimental process is shown in Table 3.

Table 3 Data of experiment

Average speed (rpm)	Average speed (rad/s)	Torque (Nm)	Output power (W)	Input power (W)	Efficiency
1498	156.8702	0.0936	14.6831	102.8389	0.1428
1397	146.2935	0.1061	15.5188	100.7401	0.154
1301	136.2404	0.1154	15.7276	96.5426	0.1629
1207	126.3967	0.1284	16.2344	94.4439	0.1719
1107	115.9248	0.136	15.7635	94.4439	0.1669
1000	104.7198	0.1526	15.9823	90.2464	0.1771
901	94.3525	0.1659	15.6512	88.1476	0.1776
807	84.5088	0.1755	14.8313	86.0489	0.1724
706	73.9321	0.1908	14.1092	81.8514	0.1724

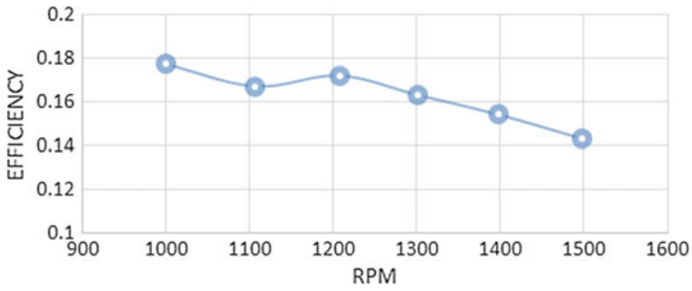

Fig. 6 Rpm versus efficiency (experiment)

Based on the results there are some consistencies from the data that has been obtained and Fig. 6 shows the data of the average speed (rpm) versus the efficiency (%).

The graph 2 shows a decrease in the result between the average speed (rpm) and efficiency. As the average speed (rpm) is increasing the efficiency will decrease.

3.3 Comparison Between Experiment and Simulation Data

The data that has been obtained from the simulation and experiment has been compared and analyzed to find the relationship between those two data. The graph below shows the combined data of experiment and simulation into one graph. Figure 7 shows the average speed (rpm) data versus the efficiency data.

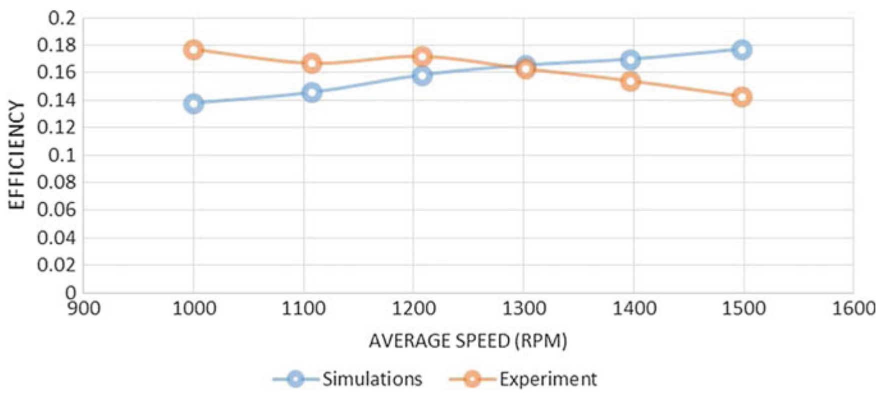

Fig. 7 Rpm versus efficiency for experiment and simulation

The compared data are supposedly to be similar or exactly with each other. Even both of data sets are not accurate, the data that was obtained from simulations and experiment are still in the accepted range. There are some errors that can be justified. One of the errors that can be justified during running the experiment is that the reading of the average speed (rpm) is not stable. This is because the reading needs some time to be stable due to the belting that is controlling the rotational speed did not have enough friction to slow down the rotating runner blade accurately that is causing the average speed (rpm) to increase back after it has been set. It required a lot of sensitive adjustment to tighten the pulley. Others errors that can be justified is the lack of computational power. Because of these factors the simulation cannot run at the most optimum meshing size as the refinement of the mashing increases the simulation time required to complete will also increase depending on the computational power.

From the references, errors that happen to other researcher are the reason for the slight difference of the computed data is due to the discretization of the domain and solution of the differential equation in the computational method and the rigidity of the computational numerical analysis than the theoretical hand calculation [8].

4 Conclusion

The experimental approach of evaluating the performance of the Francis turbine is costly as well as time consuming. Conversely, the CFD approach is faster and a large amount of results can be produced at virtually no added cost. The CFD approach for the prediction of the efficiency of Francis turbine was developed with accomplishment of analysis of the Francis turbine performance. CFD analysis shows the distribution of various parameters like the pressure, velocity 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 the 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.

Acknowledgements The authors would like to thank the Universiti Kuala Lumpur Malaysian Spanish Institute for the financial support for this research work via Final Year Project.

References

1. The Future of Hydropower in Malaysia, pp. 32–33 (May, 2005)
2. Drtina, P., Sallaberger, M.: Hydraulic turbines—basic principles and state-of-the-art computational fluid dynamics applications. **213**, 85–103 (1999)
3. Jain, S., Saini, R.P., Kumar, A.: CFD approach for prediction of efficiency of francis turbine, pp. 1–7 (2010)
4. Francis, J.B., Service, N.P.: Lowell Notes (1848)

5. Akin, H., Aytac, Z., Ayancik, F., Ozkaya, E., Arioz, E., Celebioglu, K., Aradag, S.: A CFD aided hydraulic turbine design methodology applied to Francis turbines
6. Navthar, R.R., Tejas, J., Saurabh, D., Nitish, D., Anand, A.: CFD analysis of Francis turbine. **4** (07), 3194–3199
7. Čarija, Z., Mrša, Z.: Validation of Francis water turbine CFD simulations. **50**(1), 5–14 (2008)
8. Shukla, M.K.: CFD analysis of 3-D flow for Francis turbine. **1**(2), 93–100 (2011)