

PAPER • OPEN ACCESS

Numerical prediction of hill charts of Francis turbines

To cite this article: Andreas Nordvik *et al* 2019 *J. Phys.: Conf. Ser.* **1266** 012011

View the [article online](#) for updates and enhancements.

You may also like

- [Simplified hydrodynamic analysis on the general shape of the hill charts of Francis turbines using shroud-streamline modeling](#)
I Iliev, C Trivedi and O G Dahlhaug
- [The turbulence scales of a wind turbine wake: A revisit of extended k-epsilon models](#)
M P van der Laan and S J Andersen
- [Numerical flow field investigation around guide vane of a high head Francis turbine](#)
Rohit Kumar Sahu, Bhupendra Kumar Gandhi and Shubham Sharma



ECS
The
Electrochemical
Society
Advancing solid state &
electrochemical science & technology

DISCOVER
how sustainability
intersects with
electrochemistry & solid
state science research

Numerical prediction of hill charts of Francis turbines

Andreas Nordvik, Igor Iliev, Chirag Trivedi, and Ole Gunnar Dahlhaug

Department of Energy and Process Engineering, Norwegian University of Science and Technology, Trondheim, Norway

E-mail: andnordv@stud.ntnu.no

April 2019

Abstract. This present work compares numerically predicted hill chart to experimental measurements of a Francis turbine. The main objective is to create a model for recreating hill charts using computational fluid dynamics (CFD). Accurate prediction of hill charts are useful in the design stage of production and may result in a more efficient runner. The primary focus is the prediction of efficiency and investigation of possible simplifications without loss in accuracy. By using steady-state simulations, preliminary tests were made on four different meshes, and two different turbulence models, namely the standard $k - \varepsilon$ model and the shear stress transport model. Simplifications of geometry have been tested to investigate if the simulation time can be reduced without sacrificing accuracy. Numerical simulations of 132 operating points were carried out. The efficiency was predicted with the maximal difference from measured values of 6.93%.

1. Introduction

With the rapid development of computer technology, computational fluid dynamics (CFD) has emerged as a powerful tool to directly simulate internal turbulent flow in individual or multiple components of a turbomachine [1, 2]. Turbines are tailor-made to specific conditions at a specific site, and small improvements in the geometry can have a large positive effect on operation [3]. A CFD aided design methodology applied to hydraulic turbines is, therefore, a desirable approach for increasing efficiency [4]. This has also been studied in [5, 6]. The efficiency diagram, also called hill chart, provides useful information about a turbine. The first numerically predicted hill chart using CFD was published in 1996 [7]. Accurate prediction of hill charts are useful in the design stage of production and can be used in an optimization procedure which may result in a more efficient runner [8]. The main objective of this work is to create a model for recreating hill charts using CFD.

This paper seeks to investigate the efficiency of high head Francis turbines utilizing the Francis-99 turbine at NTNU as validation for the study. By means of CFD simulations, construction of 132 operating points in the hill chart are predicted. The simulations are then compared with experimental results for validation. Trivedi et al. [9] carried out experimental



and numerical studies for a high head Francis turbine at several operating points, namely best efficiency point, high load (HL), and two different operating points at part load (PL). The simulations took 90 days to complete on a cluster of central processing units (CPU), and the results showed good agreement with experiments for efficiency. The difference between the experimental and numerical results increased on moving away from the best efficiency point, with the maximum difference ($\sim 11\%$) being observed at part load. Due to discrepancies in efficiency being large together with the aforementioned transient phenomenons, it was decided to mesh independence study on part load with guide vane (GV) opening of 70%, as well as for BEP (GV opening being 100%). Steady-state simulations for 11 different guide vane openings were carried out from PL (GV opening 40%) to HL (GV opening 140%).

2. Theory

2.1. The Hill-diagram

The hydraulic efficiency in a hydro turbine is calculated as the power output divided by the available water power:

$$\eta = \frac{\omega T}{\rho g Q H} \quad [-] \quad (1)$$

where T is the torque on the runner's hub, shroud, and blades, ω is the angular velocity of the turbine runner, ρ is the fluid density, g is the gravitational acceleration and Q is the discharge (volumetric flow rate) through the turbine.

The net head in equation (1) is, according to the guidelines set in IEC 60193 [10], defined as

$$H = \frac{\Delta P}{\rho g} + \frac{v_1^2 - v_2^2}{2g} + (z_1 - z_2) \quad [m] \quad (2)$$

, where v_1 and v_2 are the velocities at the inlet and outlet respectively. The last term describes the difference in elevation from inlet to outlet. The value of the pressure (ΔP) is acquired by the differential pressure, as shown in figure 1.

The Hill-diagram provides us with useful information about a turbine. The efficiency diagram is also called the characteristic diagram. It shows the turbines characteristics, or how it performs, under different operating conditions [11]. Hill-diagrams are created with dimensionless parameters so it is applicable for all turbines that are equally shaped geometrically. We can then compare with other rotating machinery, models and prototypes [11]. According to IEC 60193, to construct the Hill-diagram, the dimensionless volume flow Q_{ED} is plotted against the dimensionless rotational speed n_{ED} [10]. These parameters are given as,

$$Q_{ED} = \frac{Q}{D_2^2 \sqrt{gH}} \quad [-] \quad (3)$$

$$n_{ED} = \frac{n D_2}{\sqrt{gH}} \quad [-] \quad (4)$$

, where D_2 is the outlet diameter in meter and n is the rotational speed in rpm.

To create the Hill-diagram, one has to measure the flow, head, and torque. The guide vane opening has to be kept constant while the rotational speed is varied. This procedure is then repeated for several different guide vane openings, and by (3) and (4) one can plot the points along constant guide vane lines.

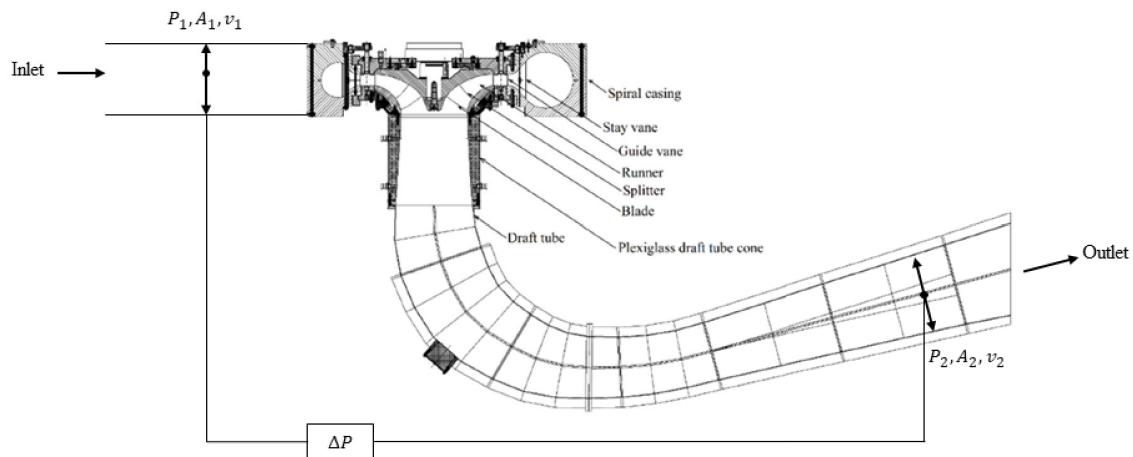


Figure 1. Two-dimensional view of the investigated model Francis turbine, retrieved from [12] and edited. $\Delta P = P_1 - P_2$ and A denotes the area.

2.2. The Francis-99 test case

The test case for this study is a model turbine at the Waterpower Laboratory at NTNU. It is a Francis type 1:5.1 scaled model of a prototype in a Norwegian power plant called Tokke [13]. It includes a spiral casing, a distributor with 14 stay vanes integrated into the spiral casing and 28 guide vanes. The runner has 15 blades with an additional 15 splitter blades, for a total of 30 runner blades. The draft tube is an elbow-type. A 2D section of the model is illustrated in figure 1. The test rig is a hydraulic system capable of generating $\approx 14\text{m}$ head for open loop, and $\approx 100\text{m}$ head for closed loop [12]. The experimental data used for validation of the CFD method in this paper is taken from [14]. However, the placement of BEP in figure 3 does not come from this study but have been tested more extensively in other measurement performed in the lab in accordance with IEC 60193 [10].

3. Numerical model

3.1. Computational domain

A complete simulation of the turbine with spiral casing, distributor, runner and draft tube including the labyrinth seal and disk friction losses is necessary for a realistic simulation of the flow in Francis turbines [15]. However, the increasing complexity and size of the

geometry will make a necessity for more cells in the computational mesh in order to get a good resolution of the flow in the simulation. This is computationally demanding and will result in longer simulation time. A full simulation with all details is therefore not feasible. The spiral case and draft tube are large and thereby requires a lot of cells. To reduce the computational cost, preliminary tests were conducted and are divided into three different cases; Case 1: Full model, Case 2: No spiral case and Case 3: Short draft tube and no spiral case. These tests showed that the relative speedup between Case 2 and Case 1 was about 1.8. Which means that reducing the geometry by excluding the spiral case was almost twice as fast to simulate as well as preserving the accuracy of the simulation. Case 3 showed an unstable behavior and did not converge. It was then decided to proceed with Case 2. The computational domain of Case 2 including boundary conditions is shown in figure 2.

3.2. Mesh

The meshes for the draft tube, runner and the guide vanes were made separately for different studies connected to Francis-99. The mesh for the draft tube was made with the ANSYS CAD module ICEM CFD. Similar meshes from previous studies had been tested for convergence [9]. However, a new mesh independence study was conducted in order to ensure mesh independence with the changes that were made.

Because simulations were going to be executed for several different guide vane openings it was necessary to create several meshes for each change in the guide vane angle. TurboGrid was used to rotate the geometry and create the mesh. The mesh was then checked for element quality and then exported for further use in CFX.

3.3. Turbulence models

Preliminary tests were conducted on two different turbulence models, the Shear Stress Transport model (SST) and $k - \epsilon$ -model. For the same turbine, the model turbine installed at the Waterpower Laboratory at NTNU, C. Trivedi et al. [9] conducted a numerical study where they used the same two turbulence models, namely SST and $k - \epsilon$. They found that $k - \epsilon$ was better at estimating the hydraulic efficiency with about $\sim 1\%$. Another study on the same turbine, the Francis-99, from Z. Yaping et al. [16] compared the standard $k - \epsilon$ turbulence model to the SST model with different outlet boundary conditions. They found that the differences between the experimental and numerical efficiency, head and torque simulated by standard $k - \epsilon$ turbulence model are smaller than that simulated by the SST turbulence model. Hence, the $k - \epsilon$ model could better predict steady-state efficiency. Again, on the same turbine, D. Jošt et al. [17] carried out a numerical study where they used SST, standard $k - \epsilon$ and zonal large-eddy-simulation (ZLES) with different inlet conditions and solvers in order to estimate the efficiency at three operating points. They found that the efficiency, calculated with CFX, yielded good agreement with the use of $k - \epsilon$ model, however, this was only so because both head and torque were wrong with about the same.

The selection of turbulence models for this paper was made from the basis of these studies. Since it is the same turbine in all three studies, it is easy to compare with the results of this article.

3.4. Simulation setup

The preliminary tests were performed on a local computer with steady-state analysis type and at runner speed equal $n_{ED} \approx 0.18$, and guide vane opening of 40%. The convergence criteria for all the simulations was set to $RMS \leq 10^{-5}$. Simulations were set to run for 1000 iterations even if the convergence criteria for rms of pressure, mass-momentum, and turbulent parameters were met.

The computational resources to perform simulations were used under the No-tur/Norstore project: Numerical investigations of a Francis turbine (project number NN9504K), using a super-computer at NTNU . No change in any parameter of interest was seen after 3500 iterations when conducting mesh independence study (set to 5000 iterations). This was done to ensure convergence for the mesh independence study. All the solution parameters used for performing the numerical simulations are shown in table 1.

Table 1. System setup for all simulations.

Parameter	Description
Analysis type	Steady state
Interfaces	Frozen rotor; discretization type-GGI
Fluid	Incompressible Newtonian fluid; water properties updated with actual density and viscosity
Boundary conditions	Inlet: total pressure inlet with direction, $P_{1,tot} = 231250$ [Pa] $\approx 12.05m$ net head Turbulence intensity 5% Outlet: Static pressure Reference pressure: 0 Pa Wall: No slip
Discretization and solution controls	Advection scheme: High resolution Turbulence numeric: High resolution
Turbulence models	Standard $k - \epsilon$
Convergence control	rms of pressure, mass-momentum, and turbulent parameters $\leq 10E-5$
Physical timescale	Auto timescale Conservative
Run type	Intel MPI Distributed Parallel: 5 nodes with 20 cores per node
Total run for GCI	PL (GV opening 70%): $n = 188$ rpm, $n = 244$ rpm and $n = 299$ rpm BEP (GV opening 100%): $n = 320$ rpm

All meshes were connected together with Frozen rotor interface between the stationary and the rotating domain. The frozen rotor interface works so that the frame of reference is changed but the relative orientation of the components across the interface is fixed. This model produces a steady-state solution to the multiple frame of reference problem, with some account of the interaction between the two frames. These interfaced together with the boundary conditions are shown in figure 2.

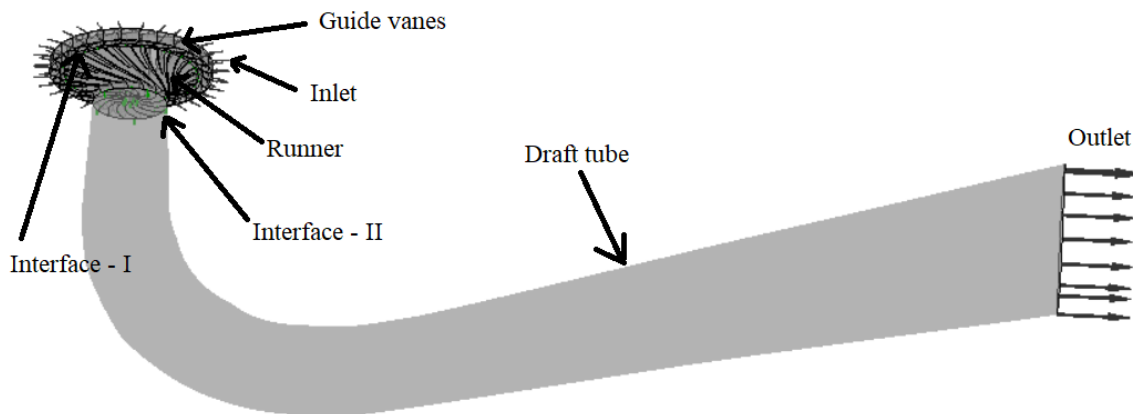


Figure 2. Computational domain of the model Francis turbine with two interfaces namely guide vane to runner (interfaceI) and runner to draft tube (interfaceII). Made similar to [9].

3.5. Boundary conditions

In addition to choosing interface and turbulence model, different boundary conditions have been tested in order to look at the effect of these. In the aforementioned study, Z. Yaping et al. [16] showed that the opening-type boundary condition gave a slightly better estimation of efficiency. In order to further investigate these effects, preliminary tests of several different outlet conditions were conducted. The pressure at the outlet was set equal to the measured pressure from experiments $P_{out} = 113kPa$.

As well as outlet condition, it is necessary to prescribe a proper inlet condition. Preliminary tests were conducted with mass flow inlet condition. However, when designing a new turbine, the flow rate corresponding to a certain guide vane opening is not known in advance. Therefore we want to know how accurate prediction is when a value of head is input data and a value of flow rate is the output of the numerical simulation. For that reason, numerical simulations were conducted with total pressure at the inlet. In this case, head becomes the input data, while a value of flow rate is a result of numerical simulation [17]. Total pressure inlet and static pressure outlet are very sensitive to initial guess, therefore the median pressure from experiments was applied. The total pressure inlet was set to $P_{1,tot} = 231250$ [Pa], which results in $\approx 12.05m$ net head.

4. Results and Discussion

The error in the quantities head, torque, and hydraulic efficiency will be regarded separately. This is to avoid that errors in head and torque cancels. Efficiency (η), given by equation (1), is proportional to $\propto T/H$, therefore it is important to look at the individual parameters to get a better understanding of the simulated efficiency. It is a common mistake to make, and could potentially lead to perfect results in hydraulic efficiency, even though both head and

torque have errors.

4.1. Mesh independence study

The recommended procedure for estimation of uncertainty due to discretization in CFD has been used to evaluate the mesh independence [18]. The grid convergence index (GCI) [18] is an industry-recognized method for assessing mesh quality but will not be repeated here. The computed flow parameters are tabulated in table 2-5:

Table 2. $n_{ED} \approx 0.13$, GV: 70%

Parameter	η [%]	T [Nm]	Q [m^3/s]
ϕ_1	84.600	618.96	0.16179
ϕ_2	84.488	615.02	0.16097
ϕ_3	83.940	586.76	0.15456
ϕ_{ext}^{21}	84.630	619.73	0.16194
GCI_{fine}^{21} [%]	0.051	0.156	0.11286
GCI_{med}^{32} [%]	0.217	0.958	0.75020

Table 3. $n_{ED} \approx 0.10$, GV: 70%

Parameter	η [%]	T [Nm]	Q [m^3/s]
ϕ_1	71.289	716.99	0.17163
ϕ_2	71.155	711.57	0.17650
ϕ_3	70.570	676.27	0.16350
ϕ_{ext}^{21}	71.34	718.18	0.16846
GCI_{fine}^{21} [%]	0.084	0.207	2.30914
GCI_{med}^{32} [%]	0.319	1.161	5.34657

Table 4. $n_{ED} \approx 0.16$, GV: 70%

Parameter	η [%]	T [Nm]	Q [m^3/s]
ϕ_1	92.495	511.07	0.14956
ϕ_2	92.434	508.62	0.14894
ϕ_3	91.961	486.40	0.14315
ϕ_{ext}^{21}	92.506	511.44	0.14965
GCI_{fine}^{21} [%]	0.015	0.091	0.07598
GCI_{med}^{32} [%]	0.097	0.693	0.59664

Table 5. $n_{ED} \approx 0.17$, GV: 100%(BEP)

Parameter	η [%]	T [Nm]	Q [m^3/s]
ϕ_1	94.586	614.50	0.18446
ϕ_2	94.506	610.32	0.18335
ϕ_3	94.050	588.35	0.17758
ϕ_{ext}^{21}	94.605	615.58	0.18475
GCI_{fine}^{21} [%]	0.0248	0.220	0.19732
GCI_{med}^{32} [%]	0.1306	1.078	0.95526

Here ϕ and ϕ_{ext} is a variable critical to the conclusions being reported and the extrapolated value. The subscript 1, 2 and 3 denotes the fine, medium and coarse mesh, respectively. GCI_{fine}^{21} is the fine-grid convergence index and is a measure of discretization error of the mesh. The apparent order, p , of the solution ranged from 3.37 to 7.44. The different mesh sizes were made according to [18], with refinement factor $r > 1.3$. For table 2-4, $r_{21} = 1.312$ and $r_{32} = 1.346$. For table 5, $r_{21} = 1.312$ and $r_{32} = 1.326$. Presented in table 3; the highest estimated numerical uncertainties in the hydraulic efficiencies were $\approx 0.32\%$ and ≈ 0.08 with fine and medium grid densities, respectively. In general, the medium mesh showed lower uncertainties particularly on hydraulic efficiency, compared to torque and discharge. The maximum uncertainty was discharge (Q) at 5.35% with $n_{ED} \approx 0.10$ and GV angle 70%, using medium mesh. For the fine grid, the maximum uncertainty was 2.3%, on the same operating point.

The GCI_{fine}^{21} was very low compared to the GCI_{med}^{32} . The converged solution with the medium grid was used for further simulations at different operating conditions, considering how computationally demanding the fine grid was.

4.2. Hill chart

Using the medium mesh from the GCI study, 132 simulations with 11 different guide vane angles were carried out. The results for efficiency and torque are presented in figure 3-4.

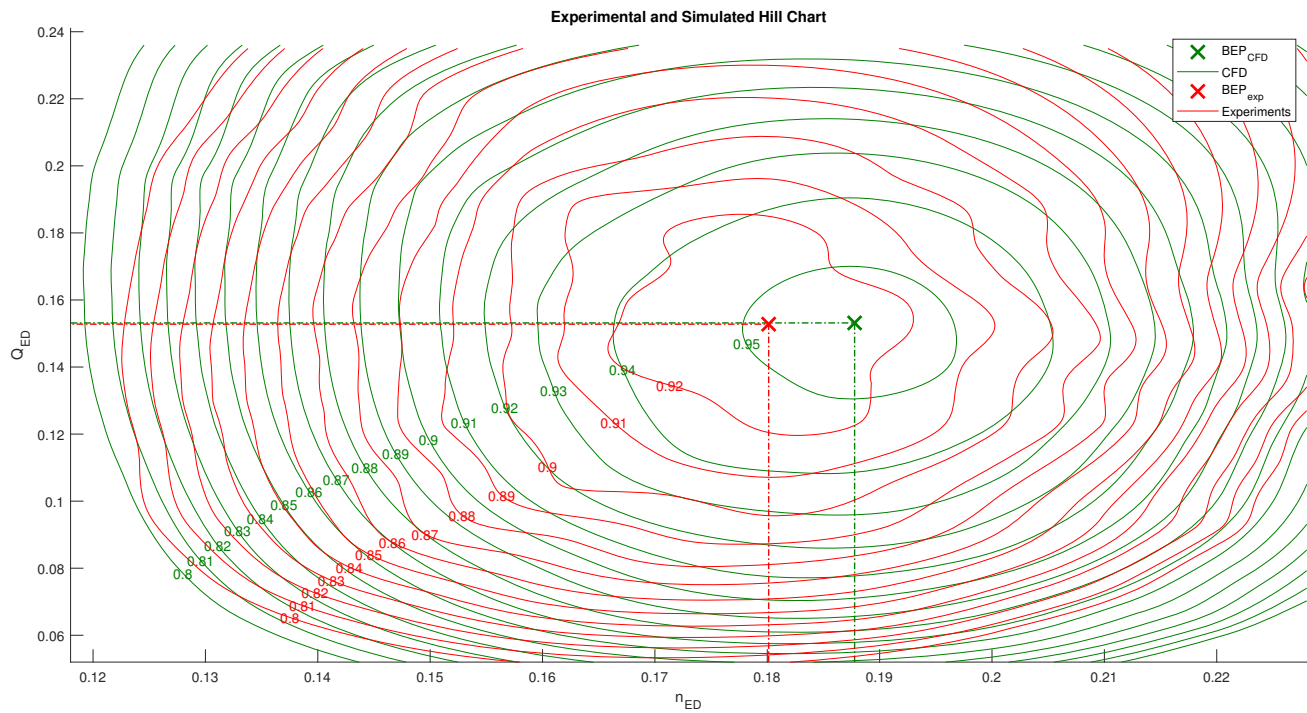


Figure 3. Red line shows experimental hill chart with data from [14]. Green line shows numerically predicted hill chart using CFD.

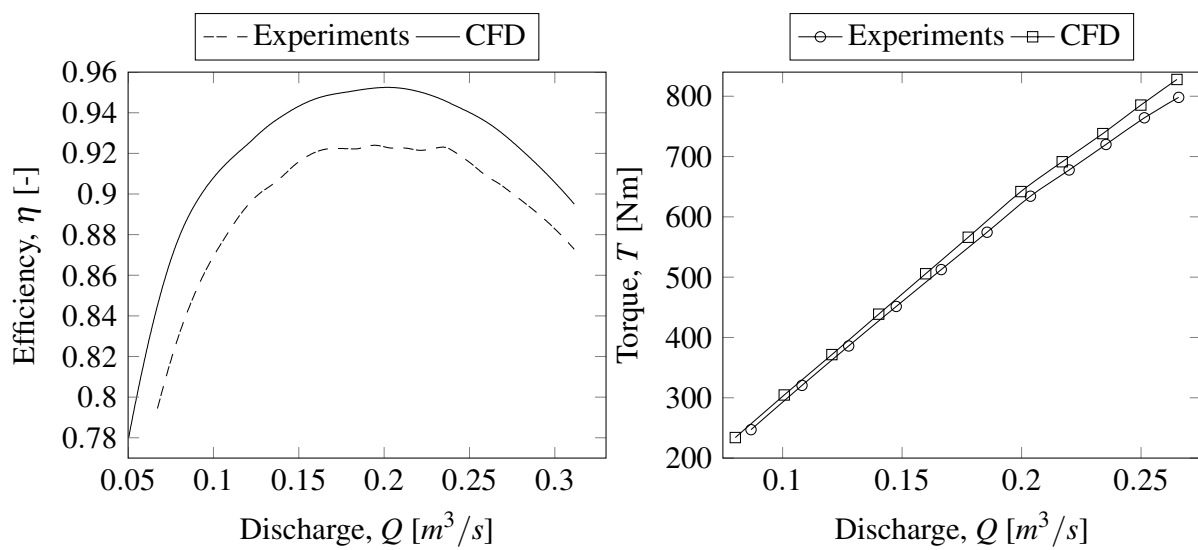


Figure 4. The left plot shows experimental and numerical CFD efficiency for constant $n_{ED} = 0.18$. The right plot shows torque for constant $n_{ED} = 0.18$.

A cross-section from the hill chart in figure 3 with constant $n_{ED} = 0.18$ is shown in figure 4. The constant n_{ED} -line describes the characteristics of normal operation. All simulations were carried out with GV opening in the range of [40%, 140%] with an increment of $\approx 10\%$ or 1° angle. The approximation ($\approx$) used throughout the paper is only used to present the data. All simulations were carried out using experimental values as input.

The hydraulic efficiency was numerically predicted with the maximum difference found at GV opening of 60% and $n = 432.5$ rpm, was calculated to be 6.93%, compared to experiments. The difference (Δ) is calculated as, $\Delta X = X_{numerical} - X_{experimental}$, where X is the variable of interest. At the lowest guide vane opening (40%), the minimum difference in efficiency was found at 0.13%, with $n = 244.4$ rpm. This is not trivial, as stated earlier, this is a highly problematic operating point with transient flow occurring in the draft tube. It is also recognizable considering other studies on the same runner, presented in a review paper by Trivedi et al. [12], showed a larger deviation in efficiency at part load than this study. The maximal difference in torque, found at GV opening of 140% and $n = 222.5$ rpm, was calculated to be 72.4 Nm.

Even more important than absolute value of efficiency is getting the shape of the efficiency curve and the position of BEP. Figure 3 shows that the BEP for numerical results is higher, i.e. higher efficiency, and shifted to the right in the hill chart compared to experiments. Q_{ED} matches almost exactly the experimental value whereas the n_{ED} is off by 4.3%. The average difference in efficiency was 2.87% for the whole hill chart. D. Jošt et al. [17] tested both head and volume flow as input data, and found that volume flow was slightly underpredicted when using head as input data. Similarly, this study underpredicted volume flow, on average, -5.6 l/s. D. Celič et al. [15] studied the influence on hydraulic efficiency from labyrinth losses. They found that the difference between the experimental and numerical efficiency lowers down from 7% to 2% at PL operation. It is believed that this is the main contributor to the discrepancies in efficiency in this study. However, the discrepancies could also come from disregarding losses in the spiral case and stay vanes. In general efficiency is overestimated, otherwise, the shape of the efficiency curve is fairly well captured.

5. Conclusion

Flow in a high head Francis turbine was analyzed by using $k - \varepsilon$ turbulence model. Numerical simulations were performed at 132 representative operating points and compared with available experimental data to verify its reliability. The difference between the experimental and numerical results was on average 2.87% overprediction, with the maximum difference (6.93%) being observed at $n_{ED} = 432.5$ and GV opening 60%. The BEP for the numerical results was higher and shifted to the right, compared to experiments. Torque was well predicted for most operating points. Simulations for the mesh independence study were carried out at part load, with guide vane opening of 70% for three different runner speeds. Mesh independence was also studied at BEP. This grid scaling test showed grid discretization

uncertainties up to 5.34% in discharge, Q. Medium mesh size was selected for further use due to computational cost. The shape of the efficiency curve was fairly well captured, however more accurate prediction could be achieved with other turbulence models.

References

- [1] T. Vu and W. Shyy, "Performance prediction by viscous flow analysis for Francis turbine runner," *Journal of fluids engineering*, vol. 116, no. 1, pp. 116–120, 1994.
- [2] M. Sabourin, Y. Labrecque, and V. De Henau, "From components to complete turbine numerical simulation," in *Hydraulic Machinery and Cavitation*, pp. 248–256, Springer, 1996.
- [3] K. Anup, B. Thapa, and Y.-H. Lee, "Transient numerical analysis of rotorstator interaction in a Francis turbine," *Renewable Energy*, vol. 65, pp. 227 – 235, 2014. SI:AFORE 2012.
- [4] H. Akin, Z. Aytac, F. Ayancik, E. Ozkaya, E. Arioz, K. Celebioglu, and S. Aradag, "A CFD aided hydraulic turbine design methodology applied to Francis turbines," in *Power Engineering, Energy and Electrical Drives (POWERENG), 2013 Fourth International Conference on*, pp. 694–699, IEEE, 2013.
- [5] J. Wu, K. Shimmei, K. Tani, K. Niikura, and J. Sato, "Cfd-based design optimization for hydro turbines," *Journal of Fluids Engineering*, vol. 129, no. 2, 2007.
- [6] S. Aradag, H. Akin, and K. Celebioglu, "CFD based design of a 4.3MW francis turbine for improved performance at design and off-design conditions," *Journal of Mechanical Science and Technology*, vol. 31, pp. 5041–5049, Oct 2017.
- [7] H. Keck, P. Drtina, and M. Sick, "Numerical hill chart prediction by means of CFD stage simulation for a complete francis turbine," in *Hydraulic Machinery and Cavitation*, pp. 170–179, Springer, 1996.
- [8] E. Tengs, P.-T. Storli, and M. A. Holst, "Numerical generation of hill-diagrams; validation on the francis99 model turbine," *International Journal of Fluid Machinery and Systems*, vol. 11, no. 3, pp. 294–303, 2018.
- [9] C. Trivedi, M. Cervantes, B. K. Gandhi, and O. G. Dahlhaug, "Experimental and numerical studies for a high head Francis turbine at several operating points," *Journal Of Fluids Engineering - Trancactions Of The Asme*, vol. 135, no. 11, 2013.
- [10] International Electrotechnical Commission, "Hydraulic turbines, storage pumps and pump-turbines - Model acceptance tests," Nov. 1999.
- [11] H. Brekke, *Introduction To Hydraulic Machinery*. NTNU, March 2000.
- [12] C. Trivedi, M. Cervantes, and O. G. Dahlhaug, "Experimental and numerical studies of a high-head francis turbine: A review of the Francis-99 Test Case," *Energies*, 2016.
- [13] C. Bergan, R. Goyal, M. J. Cervantes, and O. G. Dahlhaug, "Experimental investigation of a high head model Francis turbine during steady-state operation at off-design conditions," *IOP Conference Series: Earth and Environmental Science*, vol. 49, no. 6, 2016.
- [14] I. Iliev, C. Trivedi, E. Agnalt, and O. G. Dahlhaug, "Variable-speed operation and pressure pulsations in a francis turbine and a pump-turbine," *IOP Conference Series: Earth and Environmental Science*, vol. 240, p. 072034, mar 2019.
- [15] D. Celič and H. Ondráčka, "The influence of disc friction losses and labyrinth losses on efficiency of high head Francis turbine," *Journal of Physics: Conference Series*, vol. 579, no. 1, p. 012007, 2015.
- [16] Z. Yaping, L. Weili, R. Hui, and L. Xingqi, "Performance study for Francis-99 by using different turbulence models," *Journal of Physics: Conference Series*, vol. 579, no. 1, p. 012012, 2015.
- [17] D. Jošt, A. Skerlavaj, M. Morgut, P. Mežnar, and E. Nobile, "Numerical simulation of flow in a high head Francis turbine with prediction of efficiency, rotor stator interaction and vortex structures in the draft tube," *Journal of Physics: Conference Series*, vol. 579, no. 1, p. 012006, 2015.
- [18] I. Celik, U. Ghia, P. Roache, C. Freitas, H. Coleman, and P. Raad, "Procedure for estimation and reporting of uncertainty due to discretization in CFD applications," *Journal of Fluids Engineering (Transactions of the ASME)*, vol. 130, no. 7, pp. 078001 (4)–078001 (4), 2008.