Double Regulated Mixed Flow Turbine for Small Hydro Applications – Design and Operational Experience

J. Schiffer, H. Jaberg
Prof. Jaberg & Partner GmbH
Kerscheckstraße 41
8076 Vasoldsberg
Austria

H. Benigni
HFM Institute, TU Graz
Kopernikusgasse 24/IV
8010 Graz
Austria

G. Pretis
EFG Ges.m.b.H. & Co. KG
Untere Tiebelgasse 16
9560 Feldkirchen
Austria

Introduction

The double regulated mixed flow turbine was initially developed by P. Deriaz in the 1950ies to be used as a pump turbine. The new concept with a diagonal flow path has allowed the inventor to utilise the idea of adjustable guide vanes and adjustable runner blades for the medium head operation range. This is of particular value in case of significant head variations, which are characteristic for medium head pumped storage installations. Furthermore, the additional runner blade adjustability has improved the unit performance in turbine and pump mode.
Since its first technical application known from the Niagara Falls hydropower plant, the Deriaz-type turbine has become more frequent in the subsequent decades. However, until now, only a few applications solely for the turbine operation are known, although the Deriaz-turbine can close a gap between the conventional Kaplan- and Francis-turbines – solving at the same time the problem of limited adjustability of Francis turbines. Double-regulated mixed flow turbines offer vital benefits, especially in the medium head range with H = 20 – 100 m and in the case of high flow variations. The advantage of Deriaz turbines compared to Francis turbines is the high efficiency level over a comparatively wide range of head and discharge and an extended region with limited pressure pulsations. However, these indisputable advantages are offset by the slightly higher susceptibility of cavitation and the complexity of the runner blades' adjustment mechanism.
The Deriaz-type turbine concept is especially suitable for small hydropower plants with medium head and comparably high discharge variations. Therefore, it was considered for a new power plant in St. Johann, Austria, where it has to cope with strongly variable discharge and needs to be operated all year. Thus, a turbine concept with excellent part-load behaviour was requested. Instead of using two rather costly small Francis turbines, the concept of a double-regulated mixed flow turbine was chosen as it offers the best economic operation with strongly variable discharge over the year.
Due to the tight schedule, it was necessary to carry out the CFD-based optimisation and the mechanical design process in parallel while managing the permanent exchange of information. As a result, within a relatively short development time of just 2.5 months, the hydraulic design of the double-regulated mixed flow turbine was completed. Only seven months later, at the end of 2020, the turbine was successfully put into operation and has been running since then to the customer's total satisfaction.
The present paper roughly sketches the way from the first scratch to the final turbine design. The comparison of on-site measurement data with the results of the numerical simulation is included as well.

1. Site analysis

The hydropower plant focussed in the present paper is located close to "St.Johann am Tauern" in the federal country of Styria, Austria. Via a 2.2 km long penstock made of glassfiber reinforced plastics GRP, water from the river Pöls is supplied to the powerhouse of the diversion plant. The penstock is divided into three sections with the diameters of DN 1200 mm, DN 1100 mm, and the connection diameter DN 800 mm of the turbine. While the rated discharge accounts for 1.5 m³/s, the net head was calculated with approximately H = 37 m. The flow duration curve and the expected net head given at the beginning of the project are presented in Figure 1. The basis of the curve is the daily averaged discharge obtained over several years. Thus, the daily peak values and the minimum discharge per day cannot be identified; they are presumed to be precisely the ones that are particularly pronounced in the alpine area where the powerhouse is located. The statutory regulated amount of residual water is already taken into account. Unfortunately, more detailed data is not available.

Figure 1 shows that the minimum discharge amounts to approximately 20% of the discharge at full load. Thus, the plant is operated between 50% and 100% load for half of the year. During the rest of the year, an operation at medium and deep part-loads has to be considered.
The idea was to cover the whole range of operation with just one turbine and maintain the operation over the entire year without having trouble with pressure pulsations and vibrations at part-load conditions.

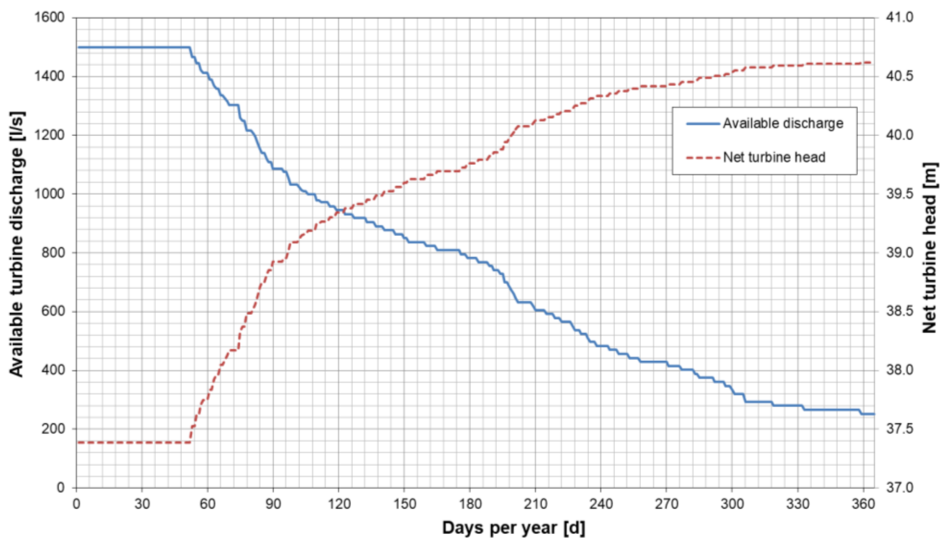

Fig. 1: Flow duration curve of the hydropower plant "St. Johann am Tauern"

Since the turbine has to be operated at partial and full loads in equal parts, a concept offering the highest peak efficiency and low sensitivity to strongly variable discharge is required. Especially for that purpose, a double regulated mixed flow turbine represents an ideal solution.

2. From an initial layout to the hydraulic design

According to the given database, the new turbine was designed for a net head of H_{Net} = 37 m and a maximum rated discharge of Q_{rated} = 1.5 m³/s. Furthermore, it shall be even capable of operating up to Q_{Max} = 1.7 m³/s for future applications. To cope with the intended suction head of H_S = + 2 m (tailwater level below reference axis of the turbine) at full load operation, the critical cavitation coefficient of the turbine needs to be below σ = 0.19 [-] (see Eq.1).

$$\sigma_{Plant} = \frac{10 - \frac{Altitude\ z}{900} - H_S}{H_{Net}} = 0.19 \qquad \text{Eq. 1}$$

It has to be pointed out that the lack of data regarding the cavitation behaviour of Deriaz-turbines impedes the selection of the overall turbine dimensions. However, Hironaka et al. [1] published a hydraulic study of Deriaz- and pump-turbines based on comprehensive model test data. According to the relation of the critical cavitation coefficient and the specific speed presented in this study, a specific speed of $n_{q,BEP}$ = 60 rpm, was chosen to achieve the cavitation target. To satisfy the equation of the specific speed (see Eq. 2), the turbine rotational speed was finally fixed with n = 750 rpm.

$$n_{q,BEP} = n \cdot \frac{\sqrt{Q_{BEP}/Q_{Ref}}}{\left(H_{Net}/H_{Ref}\right)^{0.75}} = 750 \cdot \frac{\sqrt{1.45/1}}{(37/1)^{0.75}} = 60\ rpm \qquad \text{Eq. 2}$$

Using the design recommendations given by Hironaka et al. [1], an initial meridional section and a basic overall layout of the turbine were created. Furthermore, a CFD-based optimisation of the turbine design was performed. A turbine setup with 18 guide vanes and 10 runner blades was used for the first CFD simulations. Due to the lack of space for the adjustment mechanism, the number of runner blades had to be reduced to z_{Ru} = 8 in the design process. However, the reduction of the number of runner blades increases the blade loading, and it has an unfavourable impact on the cavitation performance, which made the optimisation of the runner even more difficult.

According to the guidelines published by Hironaka and published model test data for the power plant NISHIKADOHARA [2], which was designed with approximately the same specific speed n_q, the inclination of the flow path was varied between 45° and 60°. Similar to the findings presented by Takashi Konota et al. [2], the higher inclination angle yields a higher efficiency level and a slightly lower cavitation coefficient.

Apart from achieving an excellent efficiency level and acceptable cavitation characteristics, the particular challenge was creating a runner design that integrates the runner blades' adjustment mechanism. The relatively small diameter of the spherical hub contour ($D_{Hub,in}$ = 520 mm) hardly offers enough space for the pivot bearings and adjustment levers required to operate the highly loaded blades. Thus, it was necessary to maximise the hub space while keeping the runner blades' adjustment torque at the lowest possible level.

As there are many geometrical parameters to consider, the optimisation process of a Deriaz-turbine strongly differs from the one of Kaplan- or Francis-turbines. Furthermore, fewer design recommendations are available, making finding an ideal turbine design even more challenging.

After many adaptations of the overall turbine dimensions and the shape of the runner blades, a quite compact runner design with a reference diameter of D_{Ref} = 658 mm was found. The meridional section of the final design is presented in Figure 2. While the left side of the picture shows the distribution of the meridional velocity calculated utilising CFD at the Best Efficiency Point (BEP), the right side presents the overall dimensions of the runner.

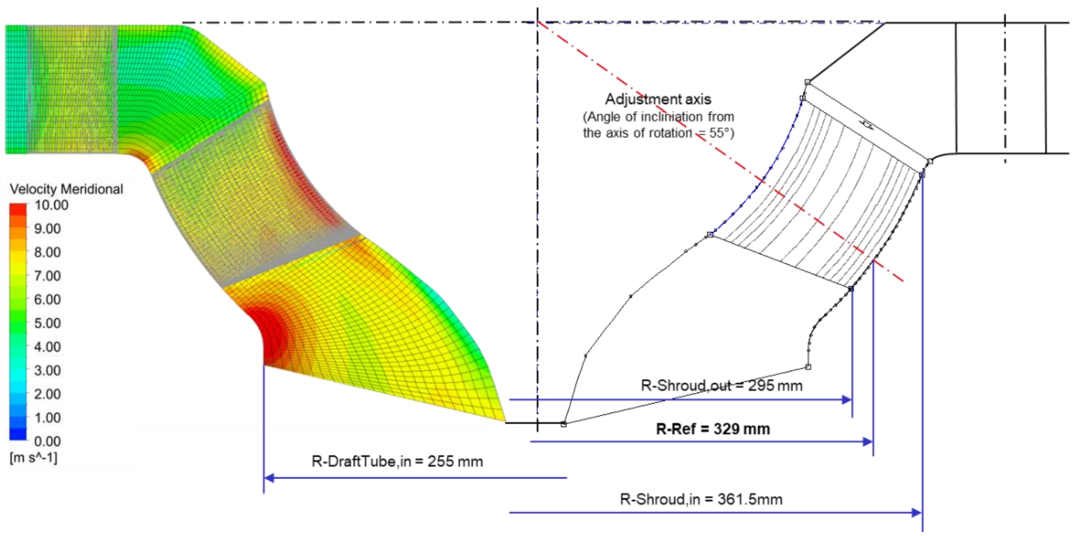

Fig. 2: Meridional section of the turbine
(left: with the distribution of meridional velocity calculated utilising CFD / right: with overall runner dimensions)

3. Numerical Simulation

To perform the numerical simulation – including the generation of meshes, the creation of the simulation setup, the solving process, and the evaluation of results – the commercial software package ANSYS-CFX V17.1© was used. CFX uses a finite volume-based discretisation scheme up to second-order accuracy and is commonly used among others for the simulation of hydraulic fluid machinery.

3.1 Description of the CFD model and the meshes used

For the preparation of the numerical simulation, the double regulated mixed flow turbine was divided into the flow domains "spiral casing", "distributor", "runner", "draft tube", and "tailwater". Then, the CFD meshes were created using the software packages ANSYS-Turbogrid© and ICEM-CFD©. An overview of the mesh sizes used for the present study is given in Table 1.

The averaged non-dimensional wall distance y^+ achieved at the best efficiency point of the turbine accounts for $y^+ \approx 25$. According to our experience, a further refinement of the meshes does not significantly improve the simulation results. A visualisation of the assembled CFD-meshes is shown in Figure 3.

Table 1. Overview of CFD meshes

Flow domain	Meshing software applied	Mesh type	Mesh size (No. of nodes)
Spiral casing	ICEM	Tetra	1.134.441
Distributor	Turbogrid	Hexa	1.399.680
Runner	Turbogrid	Hexa	2.840.672
Draft tube	ICEM	Hexa	786.150
Tailwater	ICEM	Hexa	100.556
		Total:	**6.357.387**

The meshes were assembled to either a simplified single-channel turbine model or a full 360° CFD model using ANSYS-CFX-Pre©. The single-channel model consisted of just one guide vane channel and one runner blade channel (connected with periodic boundary conditions) attached to the full draft tube and tailwater and was used to optimise the overall turbine design. In contrast, the entire 360° CFD model contained all components of the turbine – from the inlet of the spiral case to the outlet of the tailwater block – and was used for the evaluation of the final turbine design and the simulation of the operation points required to create the turbine hill chart.

The STAGE-type interface model at the transition from the guide vanes to the runner and the runner to the draft tube was used for the steady-state and single-phase CFD simulation. Thus, circumferentially averaged velocity fields were applied at the interfaces. Furthermore, as boundary conditions, a total pressure level was defined at the inlet of the spiral case. In contrast, a constant static pressure level was set at the outlet of the tailwater. Consequently, the flow rate could be adjusted according to the opening of the guide vanes and the runner blades; for the wetted walls, which were treated as hydraulically smooth, the so-called automatic (or hybrid) wall functions were used. Thus, the software automatically switched between the wall function approach and the low Re approach, depending on the grid spacing near the wall.

As turbulence model, the k-ω-based SST approach initially developed by F. Menter [3] was applied as it has repeatedly proven to achieve the best solution for engineering applications in the field of hydraulic fluid machinery in terms of robustness, stability, and accuracy (e.g. [4]). Furthermore, as the advection scheme, the "High Resolution" method was chosen for the simulations presented in this paper. Moreover, the "Automatic Timescale" option was used for the simulation process stretching over 750 iterations. With this approach, the root mean square residuals of pressure and velocity were kept below $5*10^{-5}$ and converged results for the turbine efficiency, discharge, and cavitation coefficient were achieved.

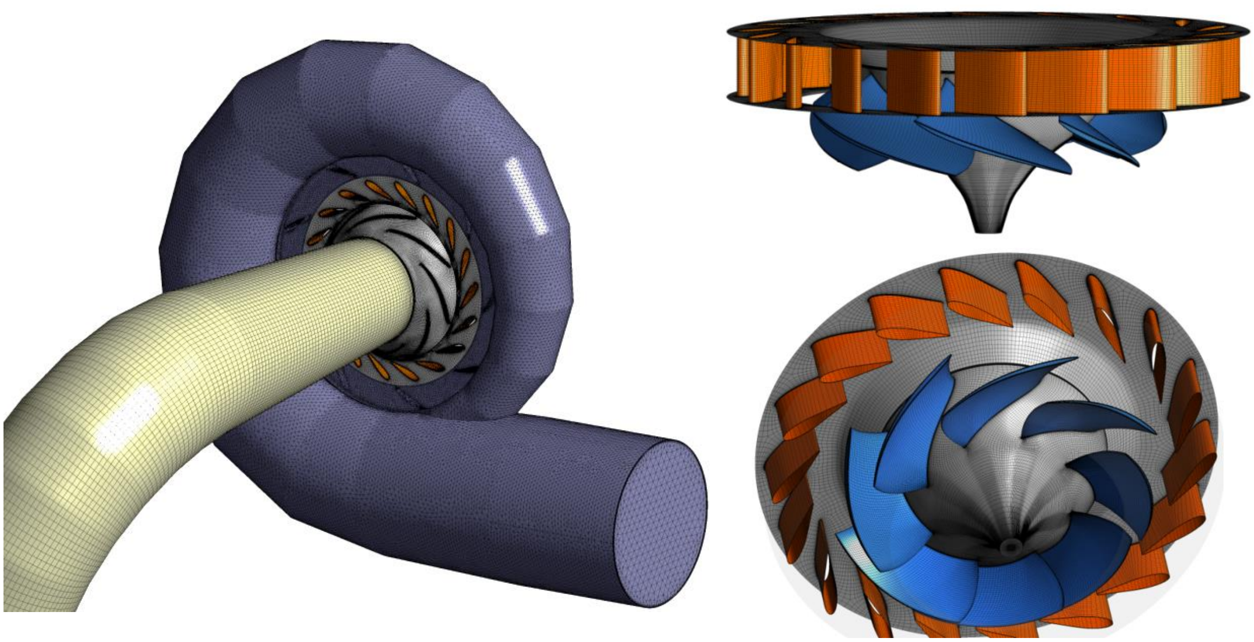

Fig. 3: Surface meshes on various components of the turbine

3.2 Evaluation of results

To evaluate the turbine performance, the IEC-definitions of the net turbine head H_{net} and the hydraulic efficiency $\eta_{hydr.}$ according to Eq. (3) and Eq. (4) was used. All quantities required for the calculations can be derived from the CFD results.

$$H = \frac{p_{Total-Inlet} - p_{Total-Outlet}}{\rho \cdot g} = \frac{\left(p_{Static-Inlet} + \frac{\rho}{2} \cdot \left(\frac{Q}{A_{Inlet}}\right)^2\right) - \left(p_{Static-Outlet,DT} + \frac{\rho}{2} \cdot \left(\frac{Q}{A_{Outlet,DT}}\right)^2\right)}{\rho \cdot g} \qquad \text{Eq. 3}$$

$$\eta_{hydr.} = \frac{\text{energetic output } P_{out}}{\text{energetic input } P_{in}} = \frac{T_{Runner} \cdot \omega}{\rho \cdot g \cdot Q \cdot H_{net}} = \frac{\left(T_{RunnerBlades} + T_{RunnerHub}\right) \cdot \frac{2 \cdot \pi \cdot n}{60}}{\rho \cdot g \cdot Q \cdot H_{net}} \qquad \text{Eq. 4}$$

To analyse the performance of each component of the turbine separately, a head loss analysis was carried out. For this purpose, the total head difference between the inlet and outlet of each component is set in comparison to the net head. Thus, Eq. (5) – the basis for the loss composition presented in Figure 4 – can be applied.

$$\eta_{hydr.} = 1 - \frac{\sum H_{Loss}}{H_{net}} = 1 - \frac{H_{Loss-Spiral} + H_{Loss-Guide\,Vane} + H_{Loss-Runner} + H_{Loss-Draft\,tube}}{H_{net}} \qquad \text{Eq. 5}$$

With the help of an appropriate statistical evaluation method, it is possible to indicate cavitation susceptibility, although only single-phase CFD simulations were performed. For this purpose, the so-called "σ-histogram method" was applied to validate the cavitation performance. The results achieved with this method were cross-checked several times with experimental data gained from the model test rig [5]. According to our experience, the σ-values calculated thereby correspond to the σ_0-values detected in the course of a model test.

4. Overview of CFD results

Based on the CFD results calculated with the initial turbine design, a stepwise optimisation of the meridional shape and the blade angle distribution was required to approach the overall performance targets.
Referring to the final turbine design and a net head of H = 36.5 m, Figure 4 presents the hydraulic turbine efficiency (orange curve) and the cavitation coefficient (green curve) calculated using CFD. The intersection of the σ-curve of the plant and the turbine highlights the cavitation limit found at Q ≈ 1.65 [m³/s]. Therefore, according to our experience with applying the "σ-Histogram method"[5] to evaluate the cavitation coefficient, a turbine operation up to the guaranteed maximum discharge of Q = 1.5 m³/s will be possible without any negative impact of cavitation on the turbine performance.

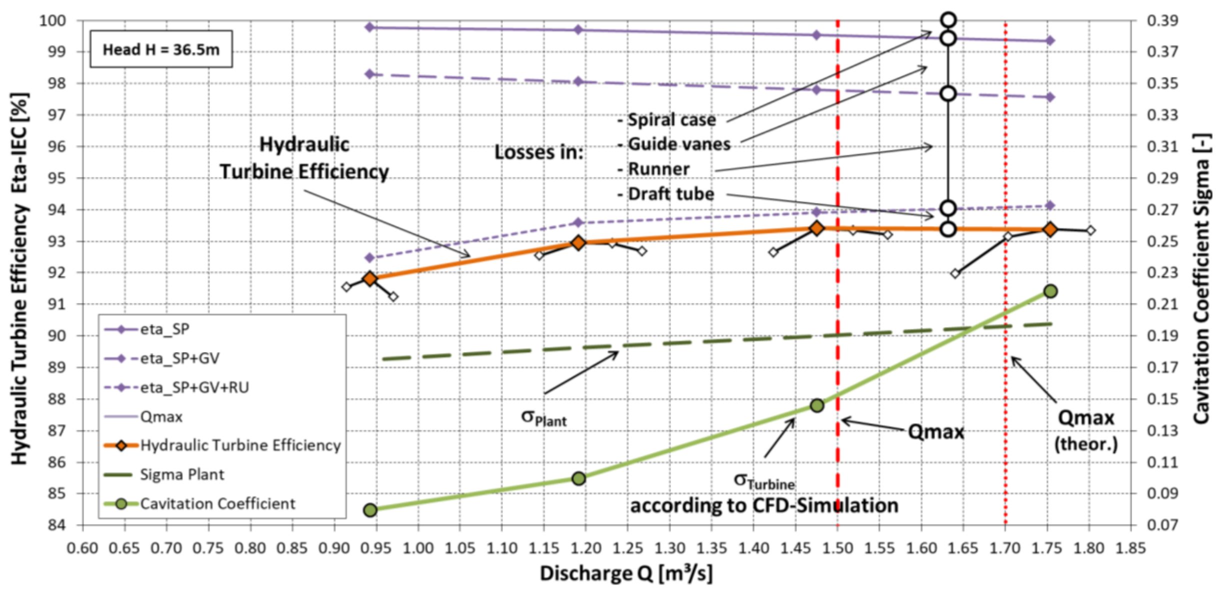

Fig. 4: Hydraulic turbine efficiency and cavitation coefficient calculated using CFD at H = 36.5 [m]

According to the CFD simulation, the hydraulic peak efficiency of the turbine accounts for 93.5%. The loss composition also presented in Figure 4 shows, that the hydraulic losses in the draft tube account for only one percentage point over the whole investigated range of operation. This can be explained by the fact that the remaining swirl at the outlet of the runner can be kept close to Zero thanks to the adjustability of the runner blades. Compared to a conventional Francis turbine, this finding yields a relatively flat efficiency curve. It allows for highly efficient part-load operation, which underlines one of the benefits of the Deriaz-type turbine.

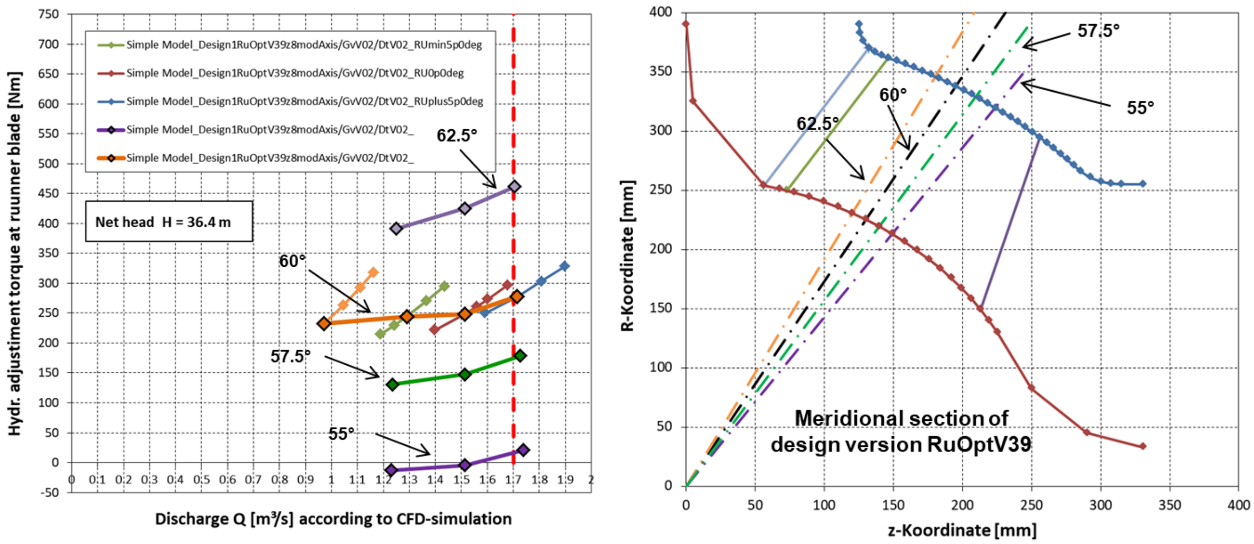

Fig. 5: Visualisation of the inclination angle of the adjustment axis (right) and its impact on the hydraulic adjustment torque of the runner blades (left)

Besides optimising the turbine performance, particular attention had to be payed to the adjustment torque of the runner blades. In the design study, it turned out that the inclination angle of the adjustment axis has the most significant impact on this critical parameter.

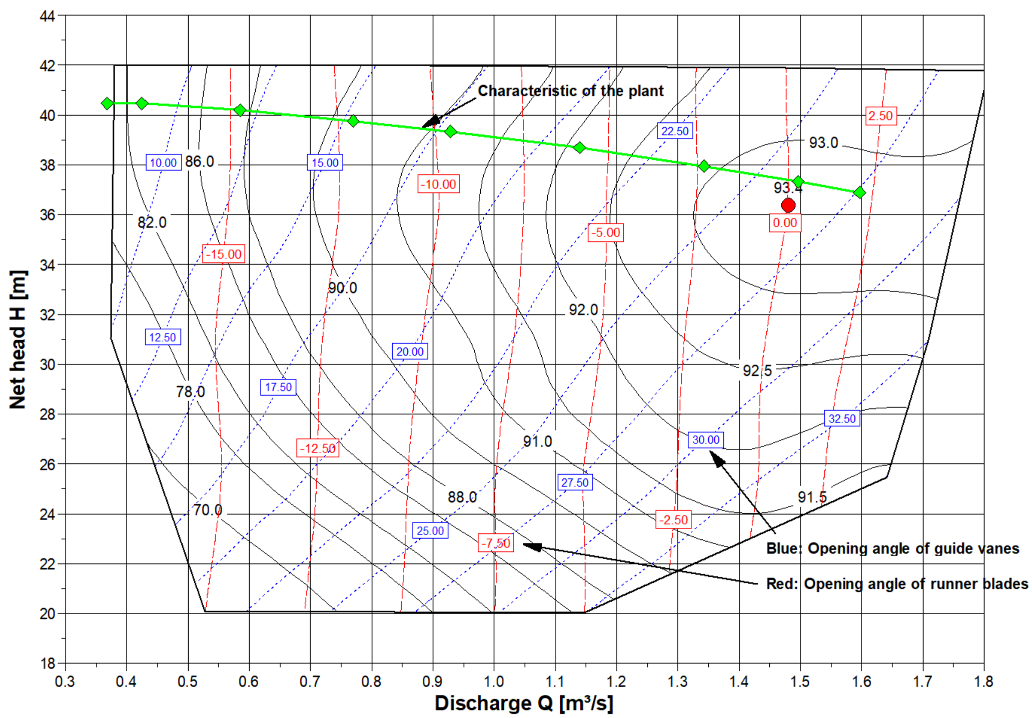

Fig. 6: Hillchart of the double regulated mixed flow turbine

On the example of an intermediate design stage, Figure 5 presents the impact of the inclination angle of the adjustment axis on the hydraulic adjustment torque of the runner blades. At some high load operation points with a discharge of Q > 1 m³/s, the inclination angle was varied between 55° and 62.5°. To minimise the runner blades' hydraulic adjustment torque, the adjustment axis was finally inclined by 55° from the axis of rotation. However, it is essential to note that the total adjustment torque contains a hydraulic and a mechanical part. In the final runner blade design, the chosen inclination angle of 55° yields a mechanical adjustment torque approximately equal to Zero. Thus, with the final runner design, the overall load of the adjustment mechanism is kept at a minimum.

Finally, the turbine performance presented for a net head of H = 36.5 m in Figure 4 was calculated for four additional head levels, allowing for creating a turbine hill chart shown in Figure 6. While the black-coloured ISO lines represent the hydraulic turbine efficiency, the blue and red dashed lines refer to the opening of the guide vanes and runner blades, respectively. The parabolically shaped green curve shows the characteristic of the plant.

5. On-site measurement and operational experience

After the design, construction, and manufacturing processes were completed within nine months, the newly developed double regulated mixed flow turbine was taken into operation at the end of 2020. A plant measurement campaign was realised after a few months of operation to obtain proof of the performance. Since an accurate measurement of the discharge was not possible, the measurement campaign could not be carried out precisely according to IEC60041 [6]. Therefore, the discharge was determined through an indirect method instead of a discharge measurement method according to the IEC standard. By measuring the differential pressure at two positions in the spiral case (see marking crosses in Figure 7), a data-reconciliation with the results of the CFD simulation was carried out. At each investigated operation point of the turbine, the discharge was estimated as a function of the measured differential pressure (see Figure 8).

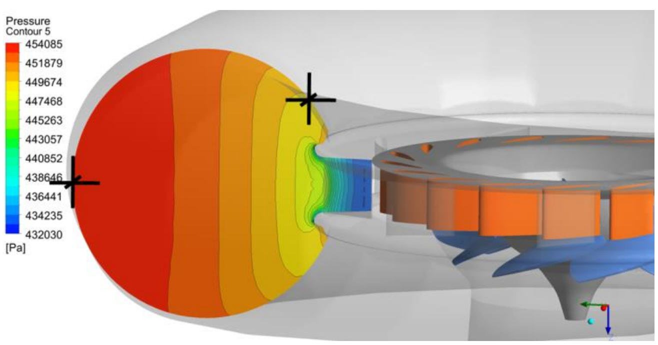

Fig. 7: Pressure distribution at a cross-section in the spiral case with two measurement positions

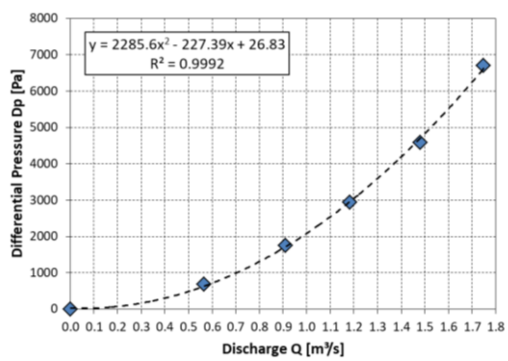

Fig. 8: Correlation of the measured differential pressure in the spiral case and discharge

Another pressure sensor was used to determine the net head available between the inlet of the spiral case and the outlet of the draft tube. The mechanical power output available at the turbine's shaft was calculated by the measured electrical power output at the generator multiplied by the generator's efficiency, which also includes the bearing losses. Additionally, the tailwater level was recorded to estimate the plant's cavitation coefficient as a function of the discharge. The openings of the guide vanes and runner blades were finally determined by using an angular position transmitter.

A comparison of the turbine efficiency measured at the plant and calculated by applying a full 360° CFD simulation is presented in Figure 9. In the case of the measured values, a typical overall error band of +/– 1.5% was considered. The comparison illustrates that the efficiency predicted through CFD was clearly exceeded in the measurement campaign. Especially in the medium part-load range, the measured performance is significantly higher than expected. However, the overall shape of the efficiency curve calculated based on the simulation results looks quite similar to the one presented for the optimised runner design of HPP Nishikadohara [7].

Furthermore, the presented results show that the prototype efficiency slightly drops at high load operation with Q > 1.5 [m³/s]. The drop in efficiency is accompanied by a rising noise level which can be associated with pronounced cavitation. This finding goes hand in hand with the cavitation limit at Q ≈ 1.6 [m³/s] predicted through CFD (see Figure 10).

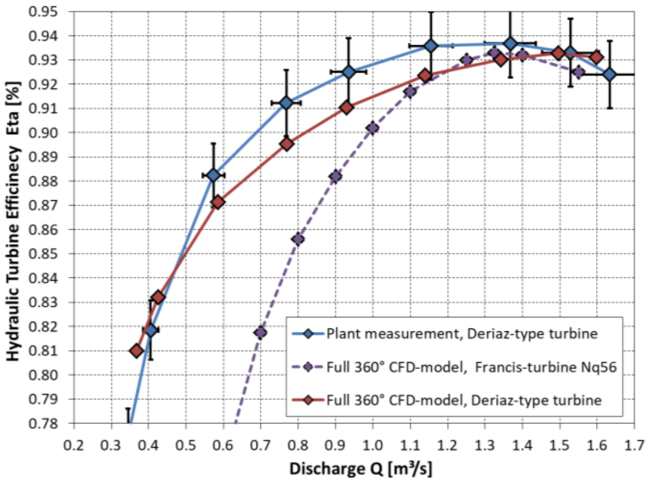

Fig. 9: Comparison of the efficiency measured at the plant and calculated through CFD

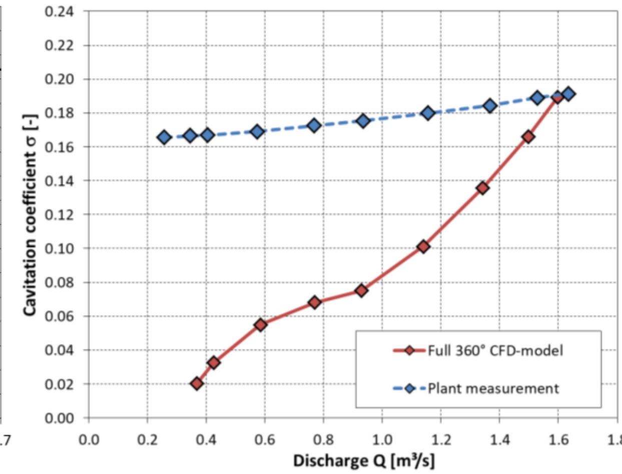

Fig. 10: Cavitation coefficient of the plant and the turbine

The purple dashed curve shown in Figure 9 represents the efficiency of a recently simulated Francis-turbine with a specific speed of n_q = 56 rpm designed for similar operation parameters. While the peak efficiency is more or less equal to the one of the new Deriaz-turbine, the efficiency significantly drops at part load operation. The well-known vortex formation in the draft tube can explain this effect, which results in increased hydraulic losses, pressure pulsations in the draft tube, and intense vibration levels [8]. In contrast, the operational experience with the newly developed double-regulated mixed-flow turbine shows that it can be operated between 15% and 100 % load without any noise emission or machine vibration problems.

In parallel with the performance measurement, the correlation of the opening of the guide vanes and runner blades was optimised. As a function of the discharge, the following diagrams compare the ideal position of the guide vanes and runner blades found by the CFD simulation and the measurement, respectively. Over a wide range of operations, the CFD results closely match the results obtained during the field tests. Only in the region of full-load operation with a discharge of Q > 1.5 m³/s a slight deviation from the ideal opening positions was necessary to reduce the noise level due to incipient cavitation.

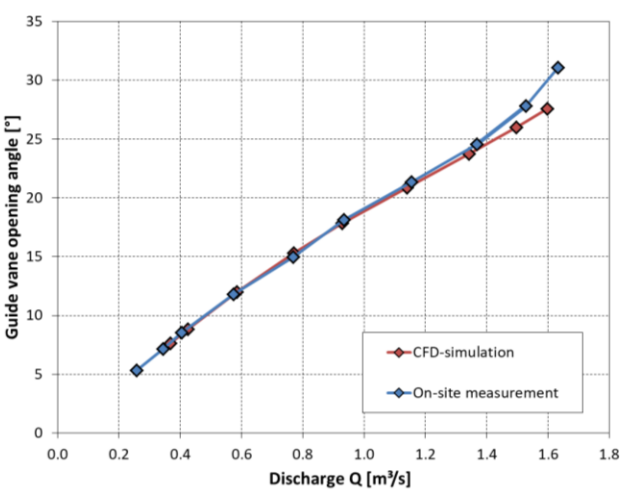

Fig.11: Comparison of the ideal guide vane opening: On-site measurement vs CFD

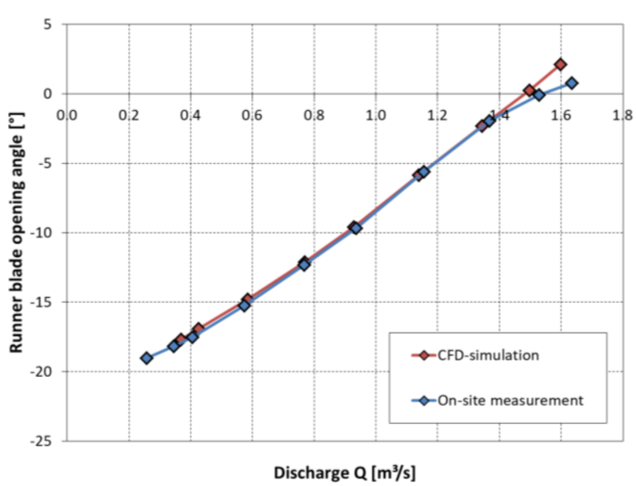

Fig.12: Comparison of the ideal runner opening: On-site measurement vs CFD

In addition to the performance test, shut-off measurements were performed in the course of the field tests. At 30%, 50%, 60%, 75%, and 100% load, the generator was disconnected from the grid, and the guide vanes were simultaneously closed. At the same time, the change of the rotational speed of the runner was recorded. According to this procedure, the highest runaway speed amounted to n = 1265 rpm, approximately 1.7 times higher than the nominal rotational speed. This value is significantly lower than the multiplier of "2" given for Deriaz-turbines with a similar specific speed n_q [2].

6. Summary

The present study paper demonstrates the design process of a double-regulated mixed flow turbine for a small hydropower plant with a nominal power output of around 550 kW located in Austria. Since the power station has to cope with high discharge variations, installing one Deriaz-type turbine was preferred over two rather costly small Francis turbines. The paper roughly sketches the way from the first scratch to the final turbine design. For the optimisation of the turbine geometry, Computational Fluid Dynamics (CFD) was applied. Besides optimising the turbine performance, the adjustment torque of the runner blades required special attention. The relatively small diameter of the spherical hub contour hardly offers enough space for the pivot bearings and adjustment levers needed to operate the highly loaded blades. Thus, it was necessary to maximise the hub space while keeping the runner blades' adjustment torque at the lowest possible level. CFD simulations using a full 360° turbine model finally achieved a hydraulic peak efficiency of around 93.5%.

In order to provide proof of performance, a plant measurement was performed after a few months of operation. The comparison of on-site measurement data with the numerical simulation results showed that the prototype efficiency exceeded the predicted one over a wide range of operations. Furthermore, since the turbine was set into operation, it runs without any troubles related to noise emission or vibration to the customer's satisfaction.

Nomenclature

A	[m²]	Area	Q	[m³/s]	Flow rate
D	[m]	Diameter	R	[m]	Radius
g	[m/s²]	Gravitation constant	T	[Nm]	Torque
H	[m]	Turbine head	z_{GV}	[-]	Number of guide vanes
H_S	[m]	Suction head	z_{Ru}	[-]	Number of runner blades
n	[rpm]	Turbine speed			
n_q	[rpm]	Specific speed	η	[%]	Turbine efficiency
P	[W]	Power	ρ	[kg/m³]	Density of water
P_{Static}	[Pa]	Static pressure	σ	[-]	Cavitation coefficient
P_{Total}	[Pa]	Total pressure	ω	[rad/s]	Angular velocity

References

1. **Hironaka, K.:** Hydraulic Study of Deriaz Turbine and Pump-Turbine, in: Mitsubishi Technical Review, 1965.
2. **Konota, T.; Ueda, T.; Takahashi, M.:** Electrical Equipment for Nishikadohara No. 3 power station, FUJI Electric Review, Vol. 15, No. 1, 1969.
3. **Menter, F.R.:** "Two-equation eddy-viscosity turbulence models for engineering applications", AIAA-Journal, 32 (8), 1994.
4. **Moessinger, P.; Jester-Zuerker, R.; Jung, A.:** "Investigation of different simulation approaches on a high-head Francis turbine and comparison with model test data: Francis-99". Journal of Physics: Conference Series 579, 01/2015.
5. **Schiffer, J.; Benigni, H.; Jaberg, H.:** Cavitation in a high specific speed Kaplan pit-type turbine – Two-phase CFD simulations and experimental verification, Conference on Modelling Fluid Flow, Budapest, Sept. 2018.
6. **International Standard IEC 60041:** Field Acceptance Tests to Determine the Hydraulic Performance of Hydraulic Turbines, Storage Pumps and Pump Turbines (Switzerland: International Electrotechnical Commission), 3rd ed., 1991.
7. **Kikuta, H.; Suzuki, R.: Nakamura, S.; Anciger, D.:** A runner modernisation in a Deriaz turbine hydropower station. 29th IAHR Symposium on Hydraulic Machinery and Systems, IOP Conference Series Earth and Environmental Science 240(2):022059, March 2019.
8. **Schiffer, J.; Benigni, H.; Jaberg, H.; Ehrengruber, M.:** "Reliable prediction of pressure pulsations in the draft tube of a Francis turbine at medium and deep part load: A validation of CFD-results with experimental data ". Proceedings of HYDRO 2018, 15th-17th of October, Gdansk, Poland.

The Authors

Schiffer Jürgen, Dipl.-Ing. Dr., studied mechanical engineering at the Graz University of Technology. Between 2008 and 2014, he worked as a scientific assistant at the Institute of Hydraulic Fluidmachinery at Graz University of Technology. He finished his PhD in the mid of 2014. Afterwards, he worked in a post-doctoral position in computational fluid dynamics for applications in the hydropower and pump industry. In 2014, he joined the Jaberg&Partner consultant group.

Gero Pretis, Dipl.-Ing., studied mechanical engineering at the Graz University of Technology. In 2008, he started working for EFG as a design engineer, responsible for the mechanical development of hydraulic turbines. Since 2020, he is the Technical Director of the company.

Helmut Benigni, Assoc. Prof. Dipl.-Ing. Dr. Studies of mechanical engineering at the Graz University of Technology, specialisation in numerical simulation, dissertation in optimisation of hydraulic machines. In post-doctoral position responsible for hydraulic machine simulations through CFD methods, development of different hydraulic designs, and machine configurations. Vice-head of the Institute of Hydraulic Fluidmachinery, Graz University of Technology.

Helmut Jaberg, Prof. em. Dr.-Ing., CEO of Prof. Jaberg&Partner consultancy since 2012. Studies of aeronautical engineering in Stuttgart, Munich and Southampton, design engineer for aircraft engines, head of development in a division and director of a business unit at KSB, certified expert for water hammer and unsteady behaviour of different power plants. Head of the Institute of Hydraulic Fluidmachinery at the Graz University of Technology until 2020.