

Design optimization of a very low head power plant - Plenty of hydraulic common sense and a bit of CFD

Cazzago Davide
STUDIO FROSIO S.R.L.
Via P.F. Calvi 9
25123 - Brescia
Italy

Luigi Lorenzo Papetti
STUDIO FROSIO S.R.L.
Via P.F. Calvi 9
25123 - Brescia
Italy

Giacomo Negrinelli
STUDIO FROSIO S.R.L.
Via P.F. Calvi 9
25123 - Brescia
Italy

Introduction

In the current international context the exploitation of low head plants across existing weirs is growing in importance. The optimization of the design of the plant waterways is nowadays a necessary step in order to achieve a full exploitation of the hydroelectric potential of the sites.

One key aspect of the design optimization is the minimization of the influence on the incoming flow, in terms of turbulence and hydraulic losses, which are generated by the geometry of the plant waterways and by plant elements such as piers, steps, screens etc. Furthermore, intake swirls should be avoided because they can lead to unwanted air entrainment [Gabl et al., 2015²]. A homogeneous velocity distribution is needed to operate the turbine without the danger of vibrations [Visher et al., 1998⁵]. Unsatisfying approach flow conditions will also lead to lower turbine efficiency [Haselbauer and Kanne, 2010³]. Beside these hydraulic conditions, the protection against sediment or wood clogging has to be guaranteed [Gabl et al., 2015²]. On the other hand designers have to deal often with geometrical restrictions dictated, for example, by availability of the areas or by the length of existing weir.

Generally, cost of physical hydraulic model research is too high if compared to the price of the structure itself and to the yearly income of those type of plants; so that cannot be considered as a suitable tool for the optimization of the plant. Additional to laboratory tests CFD simulation evolves more and more to a powerful tool for examination and improvement of inlet flow conditions [Haselbauer and Kanne, 2010³] but it still remains a remarkably advanced technique. In this framework a set of repeatable and standard design solutions can be useful to obtain a low cost and functioning optimization of the plant. Some of them, which have been applied to the Les Iles case study, are presented in this paper.

Les Iles mini hydro power plant is located on the left bank of the Dora Baltea river in Valle d'Aosta across a fixed weir (Fig. 1). Once constructed it will be able to produce hydroelectric energy from a maximum discharge of 30 m³/s and a gross head of 4 m by means of a single Kaplan generating unit.

The preliminary project has been reviewed by Studio Frosio in 2015. The paper presents the specific design optimizations studied by means of standard hydraulics computations and hydraulic common sense, which were validated by means of 1D and advanced CFD simulations. Furthermore, the design solutions proposed are summarized in a set of modifications which can be repeatable in many similar situations.



Fig. 1. Footprint of the les Iles hydropower project, preliminary geometry.

1. Case study

As a first step, the critical review was focused on the correctness of the global preliminary layout in terms of presence of the fundamental components of the plant and on their correct dimensioning. Furthermore, since water has to change direction several times so that the approach flow will not be ideal, CFD simulations were performed in order to analyze the flow conditions in the inlet and outlet channel with the original geometry. A general review of the turbine scheme was also performed in order to propose multiple choices to the client and consider more compact solutions. Evaluation of head losses was done by means of 1D simulations modeling the critical points of the plant.

1.1 Fundamental components

Lack and ill design of fundamental components was detected in the rather poor preliminary design. The most important issues are discussed below.

INADEQUACY OF FLUSHING DESIGN: especially in presence of a fixed weir, the flushing gate is a fundamental component to ensure the correct operation of the intake structure in the course of time, mainly after flooding events. Considering a fixed weir of about 60 m width, a flushing gate of 1 m width was definitely too small to ensure sediment flushing in front of the intake. No exhaustive guidelines are available in the technical literature for the design of these devices [Todeschini et al., 2010⁴], nevertheless the common practice is to set the gate width around 10-15% of the total weir width. The proposal was to widen the gate to 3-5 m width. A basic 2D depth-averaged CFD simulation was performed in order to have a general idea about the area affected by scouring effects. The simulation was performed considering a discharge of 25 m³/s passing through a 5 m width flushing gate. Considering the simulation results and the Dora Baltea river characteristics in terms of sediment transportation, exceptional sediment cleaning procedures would be necessary during the operational life of the plant.

Moreover, the adoption of a mobile weir was proposed. Although this solution was considered the best technical configuration, it was abandoned due to authorization issues.

INACCESSIBILITY AND POSITIONING OF COARSE SCREEN: even if not required by the turbine manufacturer, a coarse screen at the intake can help protect the structure from floating material and wood. In the Les Iles project, a coarse screen was positioned at the very beginning of the intake, quite close to the intake gate. Maintaining that position, the cleaning procedure would be difficult to perform since accessibility for heavy vehicles hasn't been taken into account. Considering the geometry constraints and taking into account that the coarse screen is not essential for the plant, Studio Frosio's recommendation was simply to eliminate the screen, thus reducing maintenance and costs.

ABSENCE OF FINE SCREEN AND TRASH RAKE: turbine manufacturer specifications often include the presence of a fine grid to screen the turbine from floating materials. Spacing of bars depends on the runner diameter but in general it should be less than 60 mm. Such a fine spacing generally requires the installation of automatic trash rake. In the preliminary design, the fine screen and trash rake were not contemplated.

The installation requires a rectangular canal stretch at least 2-3 m long where free surface flow occurs. The section of the screen is generally designed according to the velocity of the flow through the grid, which should be less than 1 m/s. The first considered solution was to install the screen at the very beginning of the waterway ramp even if that positioning is critical for the installation of the grid itself and for the trash rack system. Furthermore, since the grid is very close to the turbine, and the main design parameters such as the approaching velocity cannot be respected, negative influence of the grid on the incoming flow and the related decrease in efficiency are expected with this solution. For these reasons a second solution was taken into account considering the possibility to install a horizontal grid and a new horizontal trash rake at the inlet of the intake. The creation of an access to the area in front of the grid for exceptional cleaning was considered as well. This second solution presents the advantages of not influencing the approaching flow close to the turbine; furthermore the cleaning system doesn't require any trash disposal being the clog just moved downstream in the river. Even if the second solution was technically better than the first one, the client chose the first one due to the longer authorization path expected for the second one.

ABSENCE OF STOP LOGS: The partial obstruction of the plant waterways by means of sand and gravel transported by water is very probable due to the plant configuration and the geomorphological characteristics of the Dora Baltea river, especially during flood events. Generally this aspect is accepted, provided that it's possible to remove the material after the flood events by means of mechanical vehicles. This option is not possible in the specific case of Les Iles HPP, because the geometry of the waterways, and in particular the presence of a pronounced step between the river bed and the waterway bottom, makes it impossible to enter and clean the waterways with mechanical vehicles after flood events. A cheap and effective solution can be the installation of

stop logs (not considered in the preliminary design) at the HPP outlet (and also at the inlet). Positioning the stop logs at the outlet and closing the control gate at the inlet during the flood events would prevent the entrance of sand and gravel in the plant waterways.

1.2 Waterways geometry-CFD simulation

A simplified 2D depth-averaged CFD simulation was performed in order to better investigate the critical flow conditions in the inlet channel.

The meshed geometry presents the global system “river-intake” taking into consideration the structure geometry proposed in the preliminary design. The upstream boundary condition was imposed in a section sufficiently far from the intake in order to make the model non-sensitive to the imposed boundary condition. Intermediate piers were considered as obstacles in the domain.

In Figure 2 the velocity magnitude is presented as the result of the simulation with a flow $Q = 25 \text{ m}^3/\text{s}$ through the plant.

The simulation underlines a strong reduction of the flow area in correspondence of the inlet and a non uniform flow distribution in the cross section immediately upstream to the turbine. It is possible to ascribe this flow behavior to:

- influence of piers at the inlet;
- downstream intake corner shaping.

Considering the reduction of the effective flow area, it is important to notice how the approach velocity could be higher than the one computed considering the overall section as effective area. Consequently higher head losses can be expected (see Table 1).

In order to reduce the shadow effect of the piers, a modification of their orientation was proposed. Furthermore, a complete reshaping of the downstream intake corner was designed. The proposal was to withdraw the entire structure of about 10 m and to smooth the corner, which was rectangular in preliminary design, so as to obtain, along with the new orientation of the piers, a more uniform flow in front of the turbine.

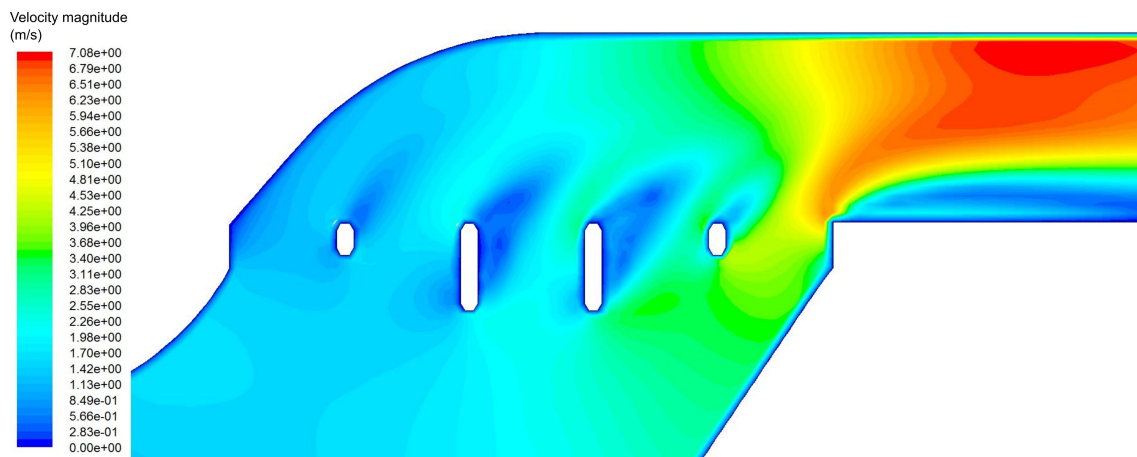


Fig. 2. Map showing the velocity magnitude (m/s) detected by CFD simulation of discharge of $25 \text{ m}^3/\text{s}$ through the intake of les Iles HPP, preliminary geometry.

A CFD simulation was performed considering the preliminary geometry also for the tailrace in order to better understand the flow behavior at the outlet. The presence of the tank and the pronounced step to reach the river bed elevation represent one of the critical points detected in the preliminary layout. Given the actual geometry a simplified 2D CFD simulation would not represent the flow behavior. For this reason a 3D CFD simulation was performed.

The modeled zone was the one between the end of the draft tube (upstream limit) and the outlet section (downstream limit). The piers at the outlet were not simulated because their influence was not crucial for the flow behavior and this allowed to reduce the computation time of the simulation.

In Figure 3 the velocity magnitude contour along the orthogonal and parallel sections of the structure is presented as the result of the simulation with a flow $Q = 25 \text{ m}^3/\text{s}$ through the plant.

From the simulation output it is clear how the effective area of outflow is a limited portion of the overall tank while the remainder is characterized by no flow conditions. Also in this case, actual velocity could be extremely higher than the one expected. Furthermore, the influence of the upstream corner of the outlet is detectable.

The proposal in this specific case was to remove the vertical step in favor of a gentler ramp. Considering the results of the 2D simulation performed for the intake, it was also proposed to smooth the upstream corner and to reshape the piers in order to limit the ineffective areas and make the flow in the outlet more uniform.

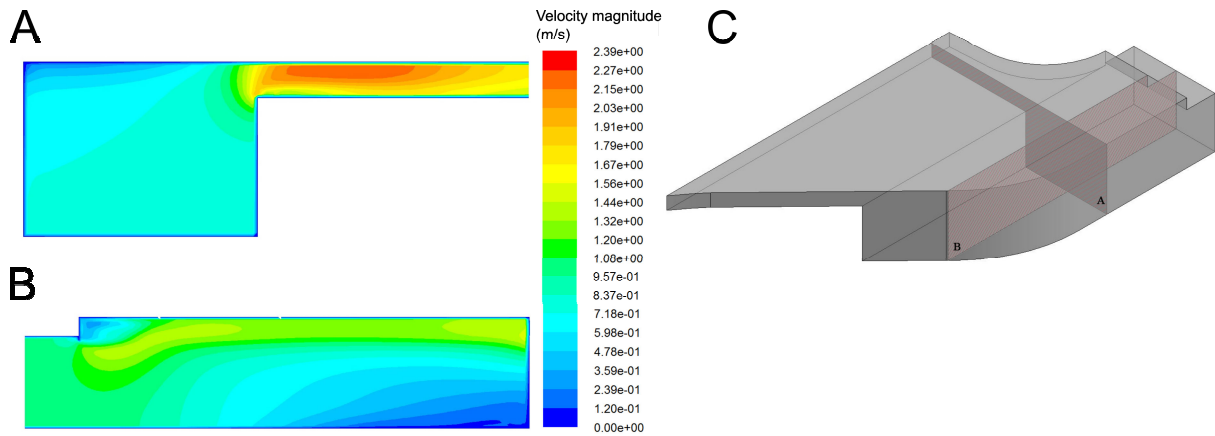


Fig. 3. Three-dimensional model of the outlet structure of les Iles HPP (preliminary geometry) and map showing the velocity magnitude (m/s) detected by CFD simulation of discharge of $25 \text{ m}^3/\text{s}$ through the sections perpendicular (A) and parallel (B) to the outlet channel.

1.3 Turbine setting

Geometric constraints and authorization impediments actually limit the overall length of the Les Iles HPP to 50 m. This is a strong limitation in terms of optimization possibilities. Essentially the limited waterways length could not ensure that the influences of the structural elements to the incoming flow were dissipated before the turbine, so asymmetry in approaching flow is detected in the section in front of the turbine. An alternative approach to extend the waterways is to reduce the length of the turbine itself. This option can be only achieved by changing the type of turbine. In the preliminary design of Les Iles HPP a horizontal Kaplan pit turbine was chosen. This type of turbines, quite common in these types of plants, ensures both the minimum depth of excavation (by reducing provisional works) and a functioning solution. On the other hand horizontal axis turbines require a greater length for the installation. This aspect is not negligible in case where, as Les Iles HPP, geometric constraints are heavy so that vertical axis turbines, having a smaller horizontal length, could be a more suitable solution. The installations of this type of turbines require deeper excavation works and more complex and expensive civil works. However, vertical axis turbines are generally cheaper than the horizontal ones so that the overall cost of the plant is not affected by the choice between the two types of turbines.

In the case of Les Iles HPP, a vertical axis Kaplan turbine solution was considered but the required geometry for the draft tube would have required a complete review of the entire tailrace canal so that the client did not accept the new solution.

1.4 Head loss computation

A one-dimensional hydraulic numerical model [Brunner, 2010¹] was performed in order to evaluate the hydraulic losses occurring in the inlet and outlet channel of the HPP. The validity of this approach is guaranteed by the fact that the simulation is related only to the specific singularity already underlined in the previous analysis. Out of this framework the one-dimensional approach cannot be considered valid anymore.

A coarse screen located at the intake of the plant and made by bars with a diameter of 200 mm and a clear opening of 300 mm was modeled. According to the two-dimensional simulations, the 1D study was carried out considering only four of the five gates concurring to the flow conveyance. Discharge through the plant varying from 15 to $25 \text{ m}^3/\text{s}$ was considered.

Figure 4 shows the output water surface profile plots for the inlet and the outlet channels. An overview of the numerical results of the model is presented in Table 1. Varying the flow rate, total head loss induced by both the inlet and outlet channels is estimated between 4% and 21% of the gross head of the hydropower plant.

The current layout geometry involves high head losses both in the inlet and outlet channel. The numerical simulation shows that flow passes from being subcritical to supercritical. The hydraulic jump resulting from the consequent supercritical flow transition represents a high energy loss. For this reason the current layout geometry is not considered suitable to convey the entire design discharge with acceptable losses.

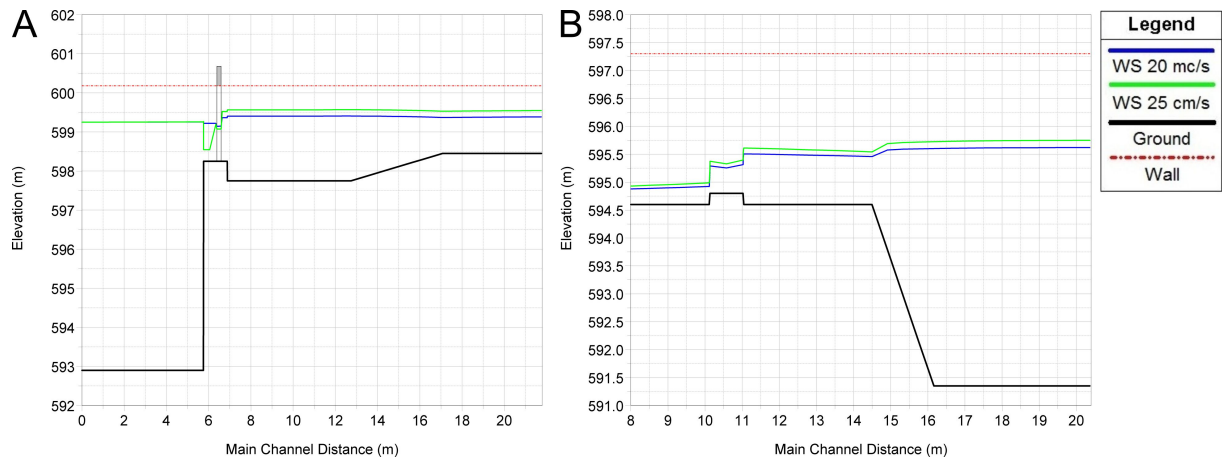


Figure 4. Output water surface profile of the inlet (A) and outlet (B) channel computed by the one-dimensional hydraulic numerical model.

Q (m ³ /s)	ΔH_{screen} (m)	ΔH_{inlet} (m)	ΔH_{outlet} (m)	ΔH_{TOT} (m)	$\Delta H_{\text{TOT}}/H_{\text{GROSS}}$ (%)
15	0.14	0.13	0.03	0.16	4%
16	0.16	0.16	0.06	0.22	6%
17	0.20	0.19	0.09	0.28	7%
18	0.24	0.23	0.11	0.34	9%
19	0.30	0.28	0.14	0.42	11%
20	0.34	0.33	0.17	0.5	13%
21	0.39	0.37	0.20	0.57	14%
22	0.44	0.41	0.22	0.63	16%
23	0.48	0.46	0.25	0.71	18%
24	0.53	0.50	0.27	0.77	19%
25	1.19	0.54	0.30	0.84	21%

Table 1. Overview of the head loss values computed by the one-dimensional hydraulic numerical model.

2. Set of repeatable modifications

Les Iles HPP presents the typical challenges of these type of plants: authorization constraints, sacrificed hydraulic design, low budget and so on. A set of modifications suitable for improving the hydraulic performance of similar plants is reported in Table 2.

Goals	Actions
Facilitate maintenance:	- widen the flushing gate - guarantee the accessibility to the screens - adoption of mobile weirs
Reduce head losses	- piers orientation parallel to the flow direction - smooth angles and hydraulic in waterways - screen as far as possible from turbine entrance section
Comply with geometrical constraints:	- vertical axis turbine - horizontal trash rake
Avoid deep excavations:	- horizontal axis turbine

Table 2. Overview of the possible actions suitable for preliminary design optimization.

2. Conclusion

The fulfillment of very restrictive geometric limits and the achievement at the same time of a good hydraulic behavior of the structure are demanding challenges; actually the best technical option often can't be realized due to external constraints.

CFD is a valid and suitable decisional instrument. But if it is used to find out quantitative results, it is very time-consuming in term of CPU time and calibration of the model. On the other hand simulations are not indispensable in this type of optimization where almost the same results can be achieved by using experience and hydraulic common sense.

Nevertheless CFD simulation can improve the relationship with the customer in supporting the different technical proposal if performed for qualitative output.

References

1. **Brunner, G.**, "HEC-RAS - River Analysis System Hydraulic Reference Manual", Davis (USA): U.S. Army Corps of Engineers - Institute For Water Resources Hydrologic Engineering Center, 2010.
2. **Gabl, R., Achleitner, S., Innerhofer, D. and Aufleger, M.**, "Adapted evaluation criterion for the velocity distribution in front of turbines", *E-proceedings of the 36th IAHR World Congress*, Vol. 28, 2015.
3. **Haselbauer, M. and Kanne, S.**, "CFD simulations as optimization tool for approach flow conditions at hydropower plants", *Proceedings of 16th International Seminar on Hydropower Plants*, TK Wien, 2010.
4. **Todeschini, S., Ciaponi, C. and Papiri, S.**, "Laboratory experiments and numerical modelling of the scouring effects of flushing waves on sediment beds", *Engineering Applications of Computational Fluid Mechanics*, 4(3), 365-373, 2010.
5. **Vischer, D., Hager, W.H. and Cischer, D.**, "Dam Hydraulics", Chichester, UK: Wiley, 1998.

The Authors

Davide Cazzago civil engineer, joined Studio Frosio S.r.l. in 2014. He is currently in charge of the numerical analysis of steady and unsteady flow in hydropower plants;

Luigi Lorenzo Papetti hydraulic and chemical engineer, involved in the design and supervision of small hydropower plants since 1990. He is currently Technical Director of Studio Frosio S.r.l.;

Giacomo Negrinelli graduated in Civil and Environmental Engineering from the University of Brescia in 2015. His final year was spent at TU München and Deltares through the Erasmus+ Programme. He joined Studio Frosio S.r.l. in 2016 and he is mainly involved in the hydrology and hydraulic design of hydropower schemes.