I Latin American Hydro Power and Systems Meeting" – 2013

CFD optimization of low head turbines intake using Fisher-Franke guidelines

Mauricio Angulo [1], Sergio Liscia; Laboratorio de Hidromecánica, Facultad de Ingeniería, Universidad Nacional de La Plata.

DESIGN AND SELECTION OF HYDRO MACHINERY

I. INTRODUCTION

Poor design of low head turbine intakes can lead to a non-uniform flow incoming to the machine inlet. This can affect performance and operational features along its useful life.

In most of the cases, turbine acceptance test are performed with ideal intake conditions without considering the actual upstream flow condition. This situation may result in different behavior between the model and the prototype, so it is advisable to model the flow approaching the inlet section. Such modeling is usually studied by physical modeling which can be costly and may require excessive construction and test time, especially when an optimized design is required, due to this fact is preferred CFD modeling.

This work aims to propose the use of a methodology that allows the optimization of low head turbine intakes. The methodology takes as its starting point the Fisher-Franke´s [1] authors guidelines.

In particular this paper shows the design optimization of a bulb intake, where the flow pattern upstream the inlet section it is not usual, so the main concern was to guarantee a high quality flow at the turbine inlet section.

II. MATERIALS AND METHODS

In order to optimize an intake design it is proposed a methodology that has its basis on Fisher-Franke´s guidelines. The proposed methodology has some differences with its precedent mainly in two aspects:

• It is aplied on CFD results and not on physical model survey.

• The criteria that indicate the wright direction of proposed designs it is not a static criteria but a criteria that depends on the particular case named as "ideal" flow condition.

According to F&F guidelines, there are some requirements that a good low head turbine intake design should satisfy. These requirements are summarized below:

Requirement 1: The normalized axial flow velocity at the intake section should fit within a certain spatial distribution named as Fisher-Franke limits (red dash lines, Figure 7).

Requirement 2: At trash rack section, no cross flow velocity should exceed +/- 5 % of mean axial velocity.

Requirement 3: Flow free from air entraining vortices.

Requirement 4: Deviation of the mean axial velocity for each quadrant should not vary more than 10 % from mean axial velocity.

Requirement 5: A deviation of the local flow velocity should not exceed 5º. This value could be exceeded only if it not results in a rotating flow.

All of these requirements are evaluated at a unique section close to the turbine inlet, named a **reference section**. The recommended section should be behind the trash racks, near to the turbine components.

The new methodology reformulates the *1st Requirement*. Instead of requesting that the axial velocity be closer to a fixed and specific velocity spatial distribution, it is proposed to get closer to a spatial distribution named as "Ideal".

To know the velocity distribution at the reference cross section, it is proposed to model an intake by means of CFD where the inlet flow distribution is perfectly uniform. To achieve this condition on a bulb turbine type, a tubular intake was modeled (Figure 5).

On bulb type turbines the reference section can be placed between the bulb and the gate slot. Probably the velocity profile is influenced by the bulb, but this is not a problem

1 – Laboratorio de Hidromecánica, Calle 47 Nº 200, La Plata, Buenos Aires, Argentina. (+54)0221-4236684, mauricio.angulo@ing.unlp.edu.ar

because the 1st requirement is assessed in comparison to the "Ideal" flow condition where the same reference cross section was computed.

On Kaplan type turbines it is recommended to take the reference section at the spiral case inlet, behind the trash racks and also behind the gates or stoplogs slots.

The *2nd Requirement* it is proposed to be evaluated behind the trash racks. The main reason to this change is that the methodology takes a unique reference cross section to simplify the simulations results postprocessing.

For the 3th requirement, the observation of air entraining vortices on physical model tests is simple, but it is not so easy to detect them by CFD. Nevertheless some efforts in this field have been done to detect swirling flows on pump stations by means of CFD [2] [Lucino, et al].

Originally, the *4th Requirement* evaluates the mean velocities for each quadrant. This requirement is modified expressing the same in terms of flow for each quadrant. Although the ratio between the quadrant mean velocity and the section mean velocity is mathematically equal to the ratio between the quadrant flow and the total section flow, it is preferred the last one, because this parameter has a more clear physical sense for hydraulic designers.

The 5th requirement stays the same as it was stated by F&F´s guidelines.

One additional requirement is established, *6th Requirement*. Here it is proposed the use of α Coriolis coefficient which is the kinetic energy correction factor. This parameter gives an idea of how far is the actual velocity profile from the uniform velocity profile. Our experience indicates that for an acceptable intake design, α coefficient should not exceed 5 % from the "Ideal" flow condition.

Therefore, the requirements of the proposed methodology can be formulated as follows:

Requirement 1: The normalized axial flow velocity at the reference section should be closer as possible to the spatial distribution named as "Ideal" flow condition.

Requirement 2: At reference cross section, no cross flow velocity should exceed +/- 5 % of mean axial velocity.

Requirement 3: Flow free from air entraining vortices.

Requirement 4: Deviation of flow for each quadrant should not vary more than 10 %.

Requirement 5: The deviation angle of flow velocity from axial direction should not exceed 5º. This value could be exceeded only if it not results in a rotating flow.

Requirement 6: The α Coriolis coefficient at the reference section should not exceed 5 % from calculated for the "Ideal" flow condition.

All these statements must be assessed at a unique cross section that must be exactly the same for each design and for the "ideal" flow condition. Besides, it is also preferred to be placed at a section where geometry changes occurs upstream of it, in order to not distort a comparative analysis.

STUDY CASE

Applying this new methodology, it has been studied the optimization of a low head power station with bulb type turbines. The powerhouse has the particularity that allows flood passage over it. This is possible operating two radial gates, one located at the spillway crest and the other at the entrance of the spillway channel (Figure 1). The flood passage function it is not secondary, such is the case that the ratio between the flooding flow and the turbine flow is approximately 7. So the powerhouse design must accomplish both functions.
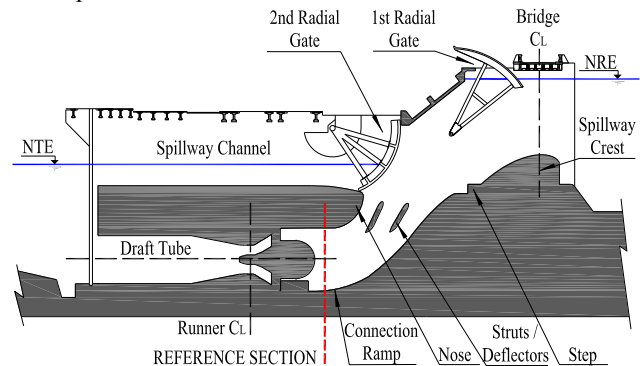

Figure 1: Longitudinal section of bulb unit.

The CFD modeling was done for one unit vane using the specialized CFD code FLOW-3D®. This code solves the Navier-Stokes and continuity equations by finite differences approximation.

1 – Laboratorio de Hidromecánica, Calle 47 Nº 200, La Plata, Buenos Aires, Argentina. (+54)0221-4236684, mauricio.angulo@ing.unlp.edu.ar

In particular this software has the ability to model transient free surface flows with high accuracy because cells can be occupied partially by fluid.

CFD MODEL SETUP

Geometry construction: The optimization process requires the intake geometry to have a flexible format that allows hydraulic designers to introduce geometry changes in a simple and quick way. For this purpose it has been used a CAD tool which is widely extended in civil works design.

The exchange format between CAD and CFD was stereo-lithography (.STL).

Four intake designs and the "Ideal" intake were modeled.

Geometry Meshing: For each design it has been used 3 orthogonal meshes composed by cubic cells. To reduce the CPU total time for each design, CFD calculations have been divided in two stages:

1st stage: model warm-up with coarse meshes.

2nd stage: model simulation with fine meshes

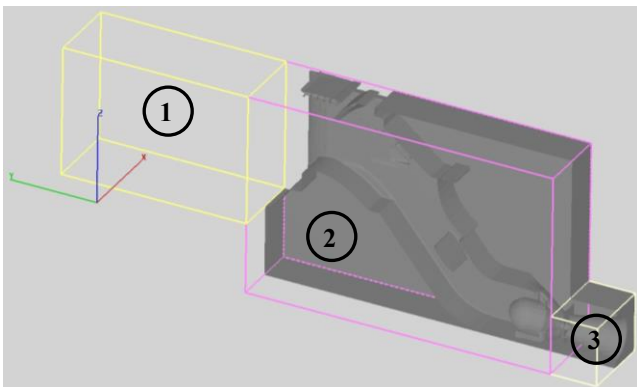| Mesh	1st Stage		2nd Stage	
	Size (m)	Cells	Size (m)	Cells
1 (Reservoir)	0,50	241.000	0,50	241.000
2 (Intake)	0,50	612.000	0,25	4.450.000
3 (Draft Tube)	0,50	27.000	0,50	27.000
Number of cells	Total	880.000	Total	4.718.000
	Active	**489.000**	Active	**1.850.000**
Simulation time (sec)	200		100	
CPU time (hs)	7,5 – 14,5 hs		52 – 60 hs	

Table 1: Geometry meshing and simulation stages

Figure 2: Geometry meshing

In all of the cases and for all the geometry components, the absolute roughness was 1 mm following the experience on previous simulations.

A baffle was added at the plane which contains the blade centerline. This baffle have porosity properties to produce a local energy loss in a such way that, given the reservoir and tailwater levels, the flow thought the turbine is equal to the nominal flow. The porosity properties of it are: porosity and linear loss coefficient are zero, and the quadratic loss coefficient is 1,75. To find this value it is necessary to iterate until the flow thought the turbine is the nominal one within a certain tolerance.

Boundary conditions: For the mesh number 1 (upstream) it was imposed the reservoir water level that corresponds with generating condition (NRE). For the mesh number 3 (downstream) it was fixed the same arbitrary constant level for each design (NTE). Summarizing, for the upstream and downstream faces for the whole domain, the boundary condition is specified pressure/fluid height and for the rest of the mesh faces a symmetry boundary condition was set. Although it is possible to impose flow condition at the upstream face, the set boundary conditions have been always resulted on stable and convergent simulations.

Turbulence Model: The turbulence model chosen was k-ε RNG. A constant maximum turbulent mixed length of 1,19 m was set. This value corresponds with the bibliography recommendations, taking 7 % of the depth at the spillway crest, as the reference length.

Initial Conditions: The initial condition for the first stage was the domain filled with fluid up to the normal reservoir elevation (NRE). The 1st stage run for 200 sec to stabilize at a flow close to the turbine nominal flow. The initial condition for the following stage was the results for the last time interval of the first stage (restart condition). These linked simulations reduce hugely CPU time when high spatial resolution is required.

Simulation: For each intake modeled it is important to know the steady state of flow. To achieve this state it is required an unknown simulation time that is defined according to the first simulation experience. The simulation time and CPU time consuming is summarized on Table 1.

Simulations were computed on a CPU with these main features: Windows 7 64-bits, Intel® Xeon® E5645 (6 cores at 2.4 GHz) and 32 GB of RAM memory.

1 – Laboratorio de Hidromecánica, Calle 47 Nº 200, La Plata, Buenos Aires, Argentina. (+54)0221-4236684, mauricio.angulo@ing.unlp.edu.ar

The flow crossing the baffle at the runner section is monitored during simulation. The criteria to stop it, establishes that flow must be between +/- 3% of nominal flow and must last, at least 50 second; that is the half simulation time for the 2nd stage.

Postprocessing: A reference cross section was defined as the methodology indicates. For each intake design modeled, the results for the last time interval of the 2nd stage are saved. For the reference section, the values of velocities for the 3 axis directions (Vx, Vy, Vz), the fraction of fluid, the volume fraction, and the position of each cell, are extracted.

Calculations are needed to evaluate the requirements; this is explained and detailed in the next section.

III. THEORY AND CALCULATION

To assess each of the requirements it should proceeded as follows:

Requirement 1:
The normalized velocity for the main flow direction (axial) must be calculated for all the reference section cells.

$$Vyn \ [-] = Vy/Uy$$

Vyn values must be in descending order. Then, it can be plotted two curves, one for Vyn > 1 and the other for Vyn < 1. The abscissa indicates the cumulated area that equalizes or exceeds a particular value of Vyn (Figure 8).

Requirement 2:
For the reference section it is calculated a histogram for each of the normalized velocities Vxn and Vzn, expressed as the percentage of mean axial velocity.

$$Vxn \ [\%] = |Vx*100/Uy|$$

$$Vzn \ [\%] = |Vz*100/Uy|$$

The histogram contains on abscissas the normalized velocity value (Vxn or Vzn) and on the ordinate the relative frequency. Figure 9 shows the Vzn histogram.

Requirement 3:
This condition was not assessed with CFD results. If the different intake designs do not modify the submergence of the bulb turbine, this requirement can be excluded of the

comparison analysis. It will only be necessary to check it for the final intake design selected as the optimum one.

Requirement 4:
To compute flow for each quadrant, the reference section was divided in 4 quadrants as indicates the next Figure.
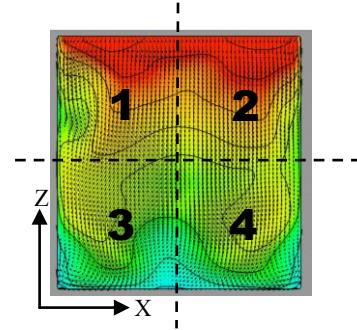

Figure 3: 4 quadrant division

Firstly the flow for each quadrant is computed: Q_1, Q_2, Q_3, Q_4 .Then the flow deviations are computed for each quadrant as follows:

$$dQ_q \ [\%] = (Q_q/Q_t) * (A_q/A_t)$$

The subscript "q" indicates the quadrant order and "t" indicates the total reference section.
The results are plotted on a graph with the quadrant order on abscissas and the flow deviation on the vertical axis (Figure 10).

Requirement 5:
For the reference cross section the histograms of deviation angles α (XY plane) and β (ZY plane) are computed.

$$\alpha \ [°] = arctan \ |Vx/Vy|$$

$$\beta \ [°] = arctan \ |Vz/Vy|$$

The histograms have the α angle on abscissas and the relative frequency on the ordinate. The same can be done for β angle (Figure 11).

Requirement 6:
The correction factor for the kinetic energy named as α Coriolis, was computed for the reference section by means of the following expression:

$$\alpha_c \ [-] = \Sigma \ [(Vyi/Uy)^3 . A_i] / \Sigma A_i$$

The subscript "i" corresponds to the cell order that forms the reference section.

If the CFD modeling is computed with FLOW-3D® code, as it is our case, it is necessary to consider the Fraction of Fluid (FF) and the Volume Fraction (VF) of each cell in order to know the actual fraction of the cells filled with fluid. This area is computed with this expression:

$$A_i = A_{cell} \cdot FF \cdot VF$$

The results are summarized on the Figure 12.

IV. RESULTS

In our study case it has been modeled the intake named as "Ideal" flow condition (Figure 5) and four different intake designs (Figure 6).
Changes introduced in the hydraulics designs modified one or more of these elements (Figure 1):
- The spillway channel nose
- The spillway step
- The connection ramp between the spillway crest and the bulb inlet.
- The strut / deflector (position and number)
Figure 5 and 6 shows the longitudinal sections for each intake. There, it could be observed geometrical changes introduced from the original design (intake 1) to the final design (intake 4).
Following, Figure 7 summarized all the reference cross sections studied. Maybe only looking at them, designers can take a decision about which is the best or which are the best designs, but this is not enough to take a decision based on quantitative and objective analysis.
Due to this reason the new methodology has been applied and the results are shown on Figures 8 to 12. With only these five graphics we have enough elements to indicate which the alternative that gets closer to the "Ideal" intake is. Also, as we are making changes we can know whether the optimization process is going in the wright way or not.
If we observe these Figures it is clear that intake number 4 is the design that fits better to the six requirements stated previously.
In respect to the 1st requirement, the normalized velocity distribution is very close to the ideal one (Figure 8).

Looking to the 2nd requirement, the intake 4, has a maximum deviation of 30 %, with a peak value at 15 %.

The requirement of reaching 100 % of cross vectors under a deviation of 5 % could be excessive and maybe impossible to achieve, taking account that for the ideal case, it cannot fulfill this condition due to the 22 % of the total cross vectors has a deviation of 10 % (Figure 9).
4th requirement, about quadrant flow distribution, is widely achieved by intake 4. It presents a distribution under +/- 5 % deviation, when the requirement is 10 %. It can also be observed that the upper quadrants take the mayor part or total flow as it is observed on the other designs (Figure 10).
For the design number 4, the 5th requirement shows an important improvement in the deviation angle. It has been only plotted the deviation angle β because it is the more significant direction. The α angle it is not affected by design changes so it was excluded from the comparative analysis. It is evident that depending on the intake geometry this angle could be relevant and must be considered in other powerhouse studies. For the intake 4, none of the vector exceeds a deviation of 20°, moreover, most vectors have a deviation of 10°. As it was observed for the 2nd requirement, it is also difficult to achieve the condition of no vectors over 5° of deviation, because it cannot be fulfilled by the ideal intake. Therefore the design was accepted because it accomplished a minimum deviation and a rotating flow cannot be observed (Figure 11).
Finally the comparison between the α Coriolis coefficient is convincing when we are talking about being closer to the ideal flow condition. For the ideal intake de α_c value is 1,01 and 1,03 for the intake number 4 (Figure 12).
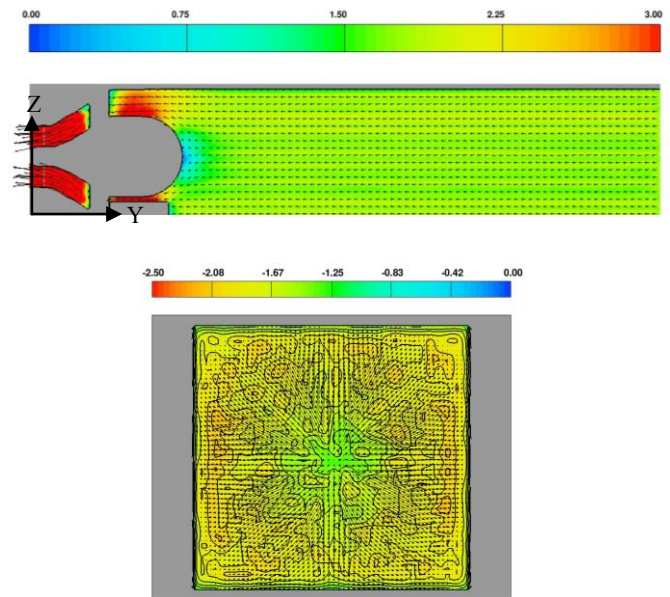

Figure 5: Ideal Intake – Longitudinal and cross sections

1 – Laboratorio de Hidromecánica, Calle 47 N° 200, La Plata, Buenos Aires, Argentina. (+54)0221-4236684, mauricio.angulo@ing.unlp.edu.ar

I Latin American Hydro Power and Systems Meeting" – 2013

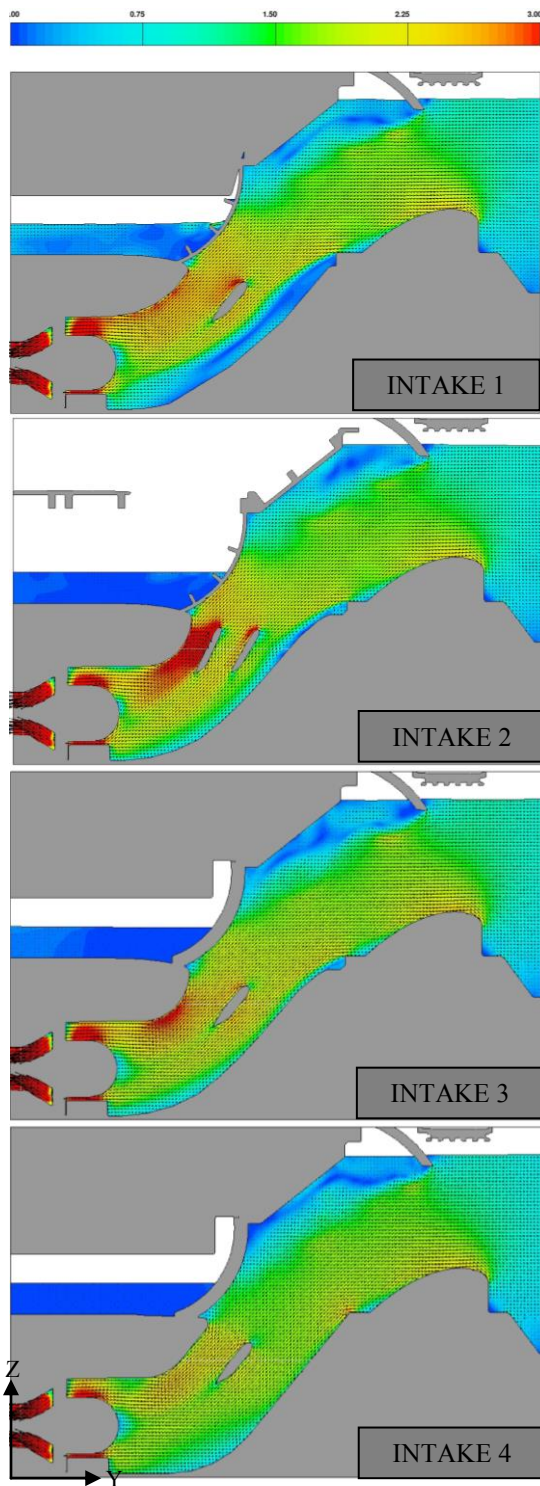Figure 6: Intake´s longitudinal sections colored by total velocity.

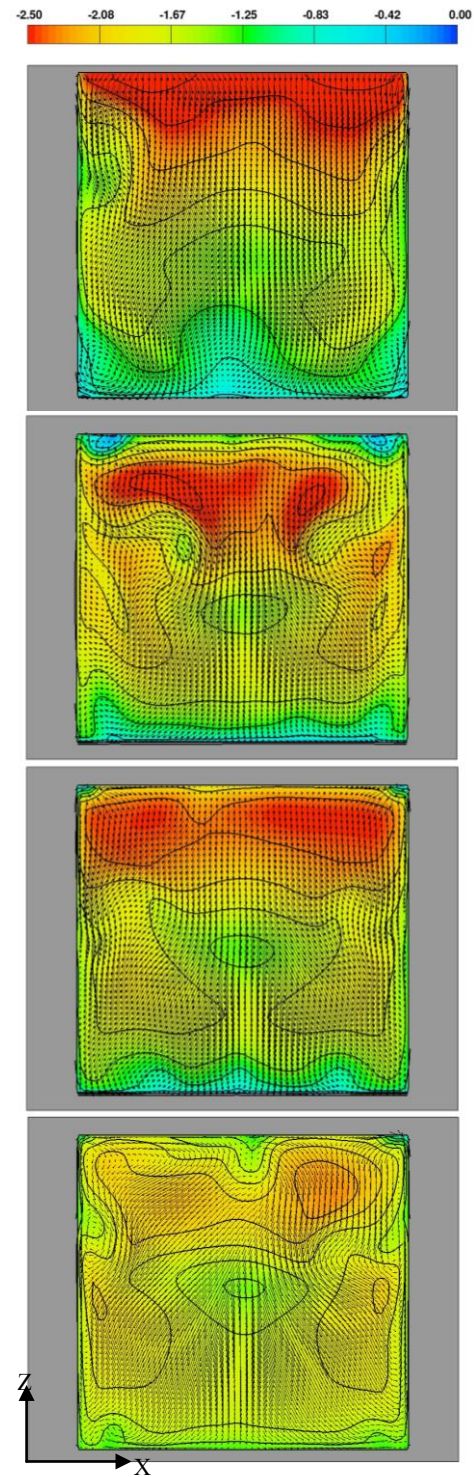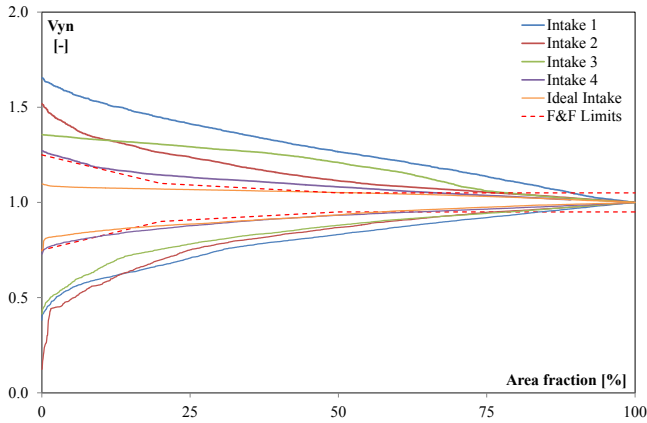Figure 7: Intake´s reference sections colored by Vy velocity

Figure 8: Condition 1 - Spatial distribution of axial normalized velocity

Figure 9: Condition 2 – Histogram of Vz deviation from axial mean velocity
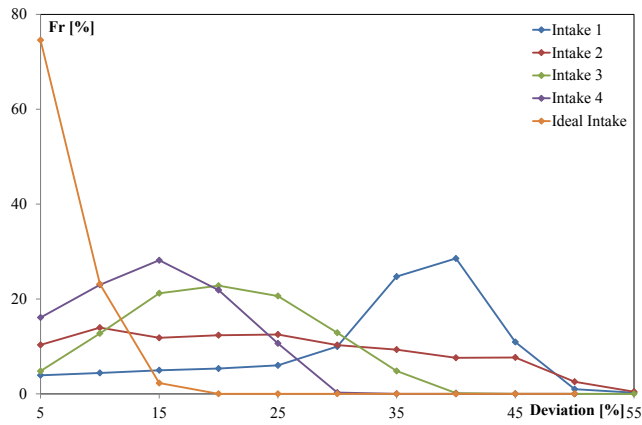

Figure 10: Condition 4 – Flow deviation for each quadrant

Figure 11: Condition 5 – Histogram of β deviation angle

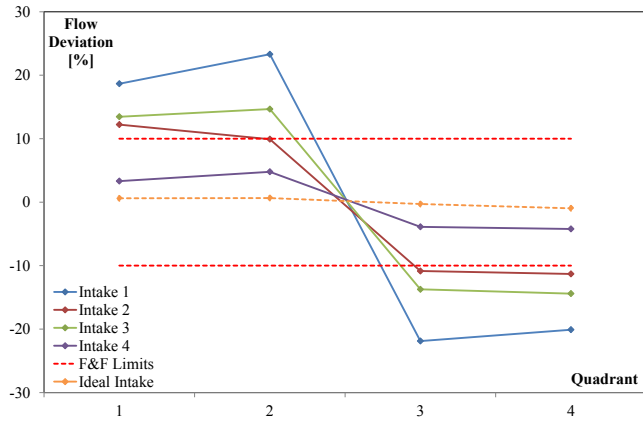Figure 12: Condition 6 – α Coriolis coefficient for each intake design.

Bellow this Figures have been repeated the more significant graphs where it can be observed a clear improvement on the hydraulic designs, starting with the intake 1 and finalizing with 4. For the 2nd requirement the histogram has its peak centered at 40 %, with a minimum of 25 % and a maximum of 50 % of deviation for the intake 1; changing to a histogram centered on 15 %, with a minimum of 5 % and a maximum of 30 % for the best design (Figure 14).

The same could be said for β deviation angle, changing from an histogram centered on 15°, with a minimum of 10° and a maximum of 35° for the intake 1, to a histogram centered on 10°, with a minimum of 5° and a maximum of 20° for the intake 4 (Figure 15).

Finally de α Coriolis coefficient for intake 1 is 1,24 and 1,03 for intake 4.
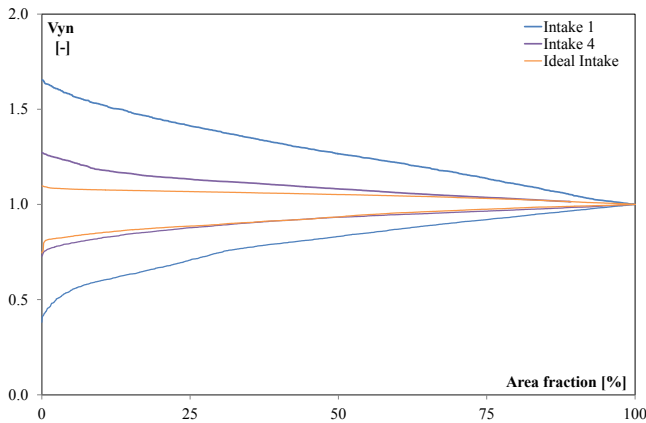
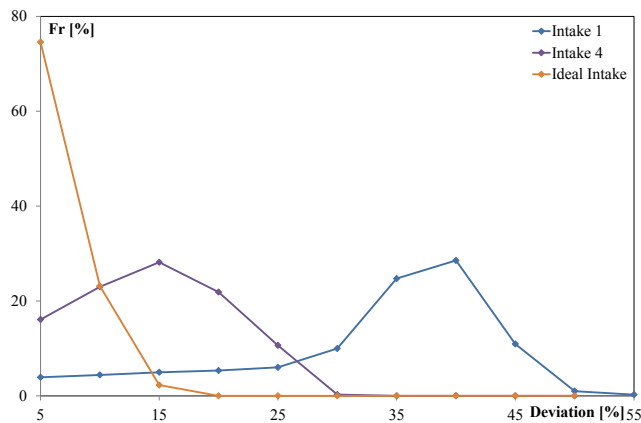Figure 13: Condition 1 - Spatial distribution of axial normalized velocity – Intake 1 and 4 comparisons

Figure 14: Condition 2 Histogram of Vz deviation from axial mean velocity - Intake 1 and 4 comparisons
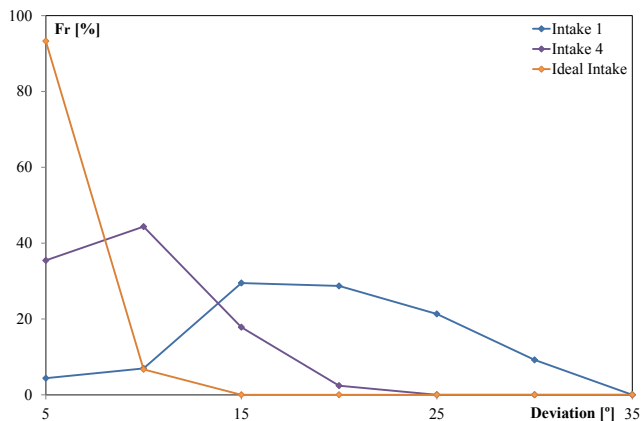

Figure 15: Condition 5 – Histogram of β deviation angle - Intake 1 and 4 comparisons

V. DISCUSSION Y CONCLUSIONS

The incorporation of the 6th requirement, suggesting the use of de Coriolis coefficient, is really important due to it is sensitive enough to detect improvements during the optimization process.

Finally it was concluded that the design and optimization process by means of numerical modeling of an intake of a low head power station beside the design evaluation with an upgraded methodology based on Fisher-Franke´s guidelines, constitute a very useful tool because it allows the designer to test their proposal quickly, objectively and with a degree of sensitivity enough to note changes on each proposed design.

At present we are working on the implementation on physical model tests to survey velocity distribution at the reference section with ADV technics. We expect to verify the design selected as optimum and we also expect to validate predictions done by CFD modeling, probing at the same time the ability of this methodology to reach the optimum design.

It is also of our interest to extend the use of this methodology in the study of the whole low head power stations. The goal will be evaluate different scenarios of power production and flood passage on isolated power station or contiguous to spillway works.

1 – Laboratorio de Hidromecánica, Calle 47 Nº 200, La Plata, Buenos Aires, Argentina. (+54)0221-4236684, mauricio.angulo@ing.unlp.edu.ar

NOMECLATURE

α : Deviation angle, XY plane.

α_C : Coriolis coefficient of kinetic energy correction.

β : Deviation angle, YZ plane.

ADV: Acoustic Doppler Velocimetry

A_{cell}: Cell área.

A_i: Effective cell area.

A_{qi}: Effective cell area for the quadrant "i".

A_t: Effective section area

CFD: Computational Fluid Dynamics

FF: Fraction of Fluid.

NRE: Normal Reservoir Elevation.

NTE: Normal Tailwater Elevation.

Vx: Axial velocity.

Vy: Vertical velocity.

Vz: Cross Velocity.

Vxn, Vyn, Vzn: Normalized velocities for three directions.

Uy: Mean axial velocity.

VF: Volume Fraction.

VI. REFERENCES

[1] Fisher, F. K. Jr., and Franke, G. F., 1987, "The Impact of Inlet Flow Characteristics on Low Head Hydro Projects . International confererence on hydropower , Porland, Oregon.

[2] Lucino, C., Liscia, S. and Duró, G., 2010, "Detección de vórtices en dársenas de bombeo mediante modelación matemática". XXIV Congreso latinoamericano de Hidráulica, Punta del Este, Uruguay.