Design of large Francis turbine using optimal methods

E. Flores[1], L Bornard[1], L Tomas[1], J Liu[2] and M Couston[1]

1 Alstom Hydro France, 82, avenue Léon Blum, Grenoble, 38041, France

2 Alstom Hydro China, Gaofeng Road, Beichen District, Tianjin, 300400, China,

E-mail: emmanuel.flores@power.alstom.com

Abstract..Among a high number of Francis turbine references all over the world, covering the whole market range of heads, Alstom has especially been involved in the development and equipment of the largest power plants in the world : Three Gorges (China –32x767 MW – 61 to 113 m), Itaipu (Brazil– 20x750 MW – 98.7m to 127m) and Xiangjiaba (China – 8x812 MW – 82.5m to 113.6m – in erection). Many new projects are under study to equip new power plants with Francis turbines in order to answer an increasing demand of renewable energy. In this context, Alstom Hydro is carrying out many developments to answer those needs, especially for jumbo units such the planned 1GW type units in China. The turbine design for such units requires specific care by using the state of the art in computation methods and the latest technologies in model testing as well as the maximum feedback from operation of Jumbo plants already in operation. We present in this paper how a large Francis turbine can be designed using specific design methods, including the global and local optimization methods. The design of the spiral case, the tandem cascade profiles, the runner and the draft tube are designed with optimization loops involving a blade design tool, an automatic meshing software and a Navier-Stokes solver, piloted by a genetic algorithm. These automated optimization methods, presented in different papers over the last decade, are nowadays widely used, thanks to the growing computation capacity of the HPC clusters: the intensive use of such optimization methods at the turbine design stage allows to reach very high level of performances, while the hydraulic flow characteristics are carefully studied over the whole water passage to avoid any unexpected hydraulic phenomena.

1. Introduction

In the recent years, Alstom has been awarded several contracts for the supply of the largest power plants in the world including Three Gorges in China (767 MW - 80.6 and 85 m), Itaipu in Brazil (750 MW – 118.4 m) and Xiangjiaba in China (812 MW - 100 m – soon in operation). Many new projects are under study to equip new dams with large Francis turbines in order to answer the worldwide electrical energy increasing demand. In this context, Alstom Hydro is carrying out many developments to answer those needs, especially for Chinese jumbo units (up to 1000MW).

The main objective of this paper is to present, using what have been learned from the previous major designs, how a large Francis turbine can be designed using specific design methods, including the global and local optimization methods. The spiral case and the distributor hydraulic profiles are designed using a methodology that increases the turbine hydraulic performances using optimized dimensions for the turbine. On its side, the runner and the draft tube are designed with a global optimization loop (involving a blade design tool, an automatic meshing software and a Navier-Stokes

Published under licence by IOP Publishing Ltd

solver) piloted by a genetic algorithm to improve simultaneously turbine performances and turbine stability characteristics.

The new optimized turbine is then model tested to confirm the expected results given by CFDOther paragraphs are indented (BodytextIndented style).

2. Background

To answer the Chinese electrical energy increasing demand, some jumbo units with Francis turbines up to 1000 MW are currently under development. For instance, the main features of the Bai He Tan Project, one of these jumbo projects, are detailed in the Table 1 hereafter.

Minimum net head (m)	163.9
Rated net head (m)	202
Maximum net head (m)	243.1
Rated output (MW)	1015
Maximum output (MW)	1015
Rated speed (rpm) to be defined	111.1 or 107.1

Table 1. Bai He Tan Project main characteristics

Such a new power plant is expected to maximize the unit output and production over the whole operating range and to decrease at the same time any unexpected rough zone or any phenomena such as inlet or outlet cavitation. The design should consider the best possible hydraulic performances but also sound turbine mechanical properties that will ensure the expected turbine lifetime. Thus, the dynamic behavior of the turbine has to be carefully studied and any periodic hydraulic phenomena that lead to a dynamic load on the turbine components must be decreased to the lowest possible value.

More specifically, the phenomena that lead to pressure fluctuations in the vane less gap and in the draft tube cone are carefully studied and any phenomena that could have an impact on the components loading are considered during the turbine design process.

At the same time, the project layout and characteristics need to be followed and the overall turbine dimensions shall be in line with the civil engineers constraints and the expected unit characteristics shall be met.

Thus, each turbine component has to be designed with the consideration of the whole turbine expected behavior and characteristics. This is all the challenge of the turbine development.

In this context, the hydraulic designer can make use of optimal methods to define a Francis turbine that meet all the client requirements. This paper presents some specific methods used to design the upstream parts of the turbine such as the spiral casing and the distributor in a first part, the runner and the draft tube in a second part

3. Spiral casing and distributor casing

3.1. Problem definition

Before going into further details, it is important to remind the function of each components of the Francis turbine casing. The role of the spiral case is to guide the flow to the stay vane inlet and distribute equally the flow through the stay vanes and the guide vanes.

The main role of the stay vanes is a mechanical one: the stay vanes ensure the link between the head cover and the bottom ring. From a hydraulic point of view, the role of the stay vanes is to feed the guide vanes with an adapted flow angle considering both hydraulic torque and optimal head losses purposes as well as a well distributed discharge (helped by the spiral case).

The guide vanes feed the runner with the proper flow velocity field and control by that way the discharge through the turbine. The modification of the guide vane opening changes the angular momentum at the runner inlet and thus allows to adjust the operating point.

This usual description of the different elements of the casing shows that :

- all the component of a turbine are highly linked.
- they address a high number of topics such as head losses and mechanical constraints.

Another important topic, for such a medium/high head designs, is the dynamic behaviour of the turbine. The rotor stator interactions, between the guide vanes and the runner blades, may produce pressure fluctuation that impact the dynamics loads on the upstream parts. These aspects have to be carefully studied during the design phase of the casing.

This is why it is important to optimize the design of the spiral case, the stay vanes and the guide vanes taking into accounts all the hydraulic, mechanical and geometrical constraints in the same time and Alstom has developed an optimal "in-house" software, that includes all these physical issues, to reach this objective.

3.2. Design of the upstream parts

To address the issues previously described, the main parameters to determine during the design phase of the casing are (see Figure 1):

- the inlet diameter of the spiral case (Dsc)
- the distance between the spiral case axis and the runner axis (Ea)
- the height of the distributor (Hd)
- the pitch circle diameter (Dp=2 x Rp)
- the radius and the angle at the inlet and the outlet of the stay vanes and guide vanes
- the length and the thickness of the profiles
- the guide vanes and stay vanes relative position and hydraulic profiles
- the runner inlet diameter (D1=2 x R1)

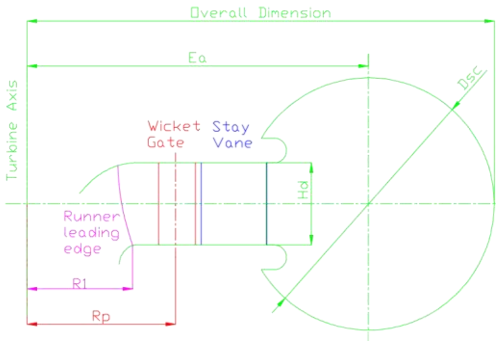


Figure 1 Main geometrical parameters for the casing definition

And all these parameters have to respect the overall dimensions coming from the civil design optimisation.

The hydraulic designer need to choose the relevant parameters to solve all the constraints, by finding the best compromise between the lowest head losses and the better dynamic behaviour while ensuring a safe mechanical design

As all parameters are linked, Alstom has developed an integrated optimal tool helping the designer to find the best solution adapted to his project. The flowchart of this tool is presented in Figure 2.

This optimal tool has been developed using several systematic CFD studies. A high number of geometries have been tested to quantify the influence of each geometrical parameter. Thanks to CFD, the flow behaviour, the flow angles, the head losses level, the guide vane hydraulic torque have been computed, analyzed and compared. Unsteady computations have been also performed to take into account dynamic phenomena such as Karman vortices at the outlet of the stay vane or the interaction between the runner and the guide vanes in the vaneless gap. Then, all this studies lead to find optimal design criteria such as kinetic momentum evolution in the distributor water passages, flow angle at the stay vanes, guide vanes and runner blades inlet, adapted distributor height and pitch circle diameter.

As mentioned above, one of the key issue of the upstream parts design are the dynamic phenomena (see reference [1]). One of them, the Rotor Stator interactions, occurs in the casing, between the guide vanes outlet and the leading edge of the runner.

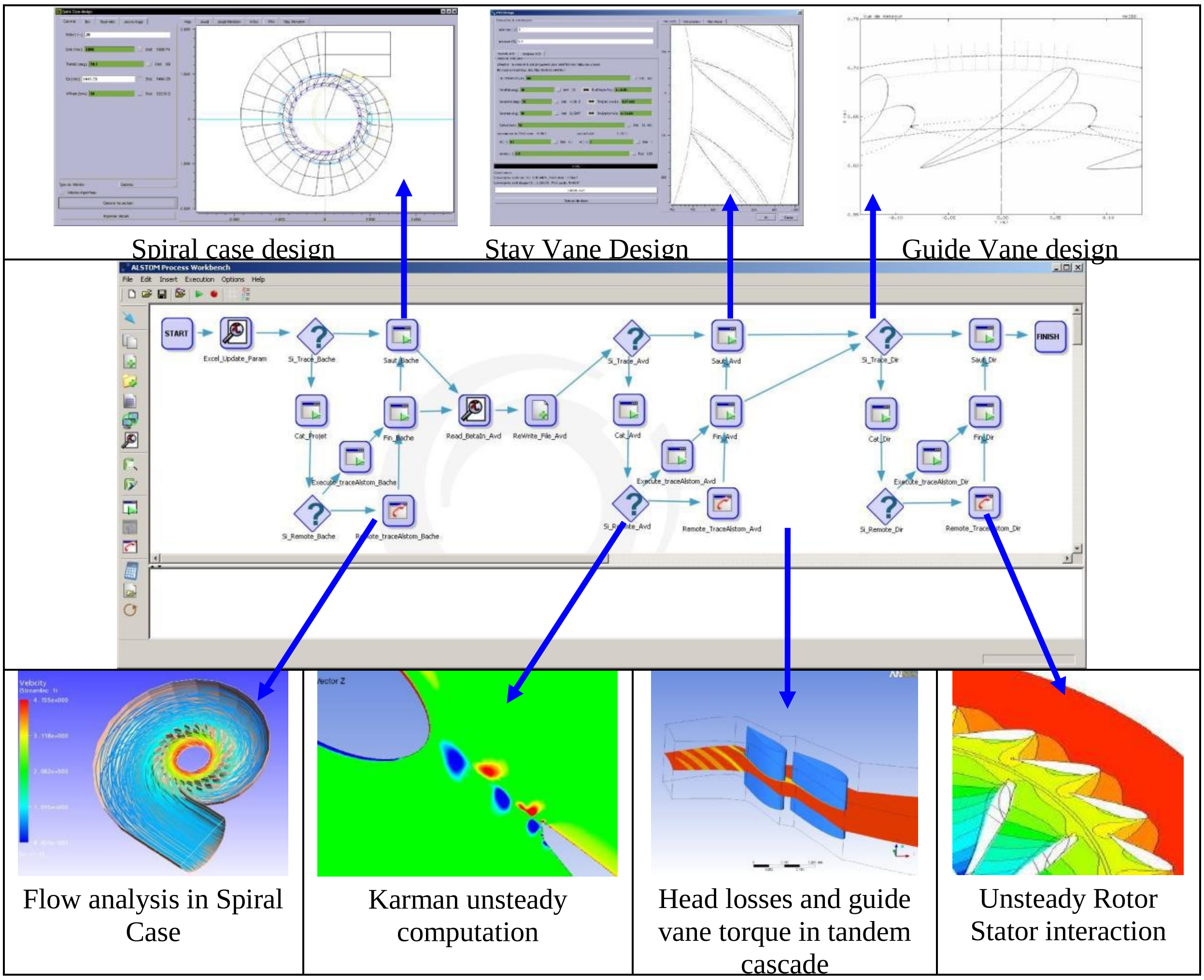


Figure 2 Optimization Tool to design the upstream parts

The pressure fluctuations in the vaneless gap are due to the rotation of runner blades in the stationary flow delivered by the guide vanes. This phenomena is mainly driven by :

- the combination of the guide vanes number and the runner blades number
- the distance between the guide vane and the runner leading edge
- the shape and the thickness of the runner blades leading edge
- the shape of the guide vane trailing edge

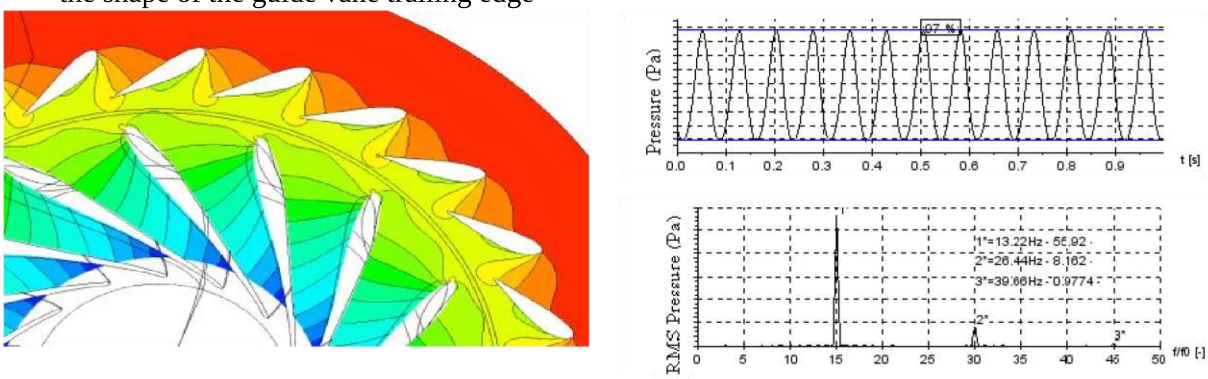


Figure 3 Numerical unsteady RSI Study : pressure field in the vaneless gap (Left)
Time and frequency analysis of a numerical probe (Right)

Thus, we can easily understand that the Rotor Stator interactions impact the casing components. Then, during the upstream parts design, all these geometrical parameters are taken into account to reduce the pressure fluctuations. Unsteady computations are performed to analyse the pressure field in the vaneless gap.

A typical numerical study of a 15 blades Francis turbine coupled to 24 guide vanes is presented in Figure 3. Typically, the pressure pulsations in the vaneless space are observed with a frequency proportional to the runner blade number. This is confirmed by the frequency analysis in the Figure 3 that shows peaks at 15*f/f0 and 30*f/f0.

This kind of computation and analysis are fully described in reference [2].

4. Runner and draft tube design

4.1. Problem definition

The civil technical requirements are considered in the determination of the distance between the distributor axis and the bottom part of the draft tube. The turbine setting is determined from the tail water level in order to prevent any risk of cavitation.

The draft tube main dimensions are determined to recover the maximum part of the runner outlet kinetic energy, within the civil allowed dimensions. These constrains guide the draft tube divergence and the flow deceleration as well as the elbow distance from the runner. The runner should deliver to the draft tube a flow velocity profile that decreases the draft tube head losses at the most representative operating conditions, and at the same time that minimizes the unsteady characteristics of the flow inside the draft tube at part load.

This explains why it is interesting to design and to compute both elements together.

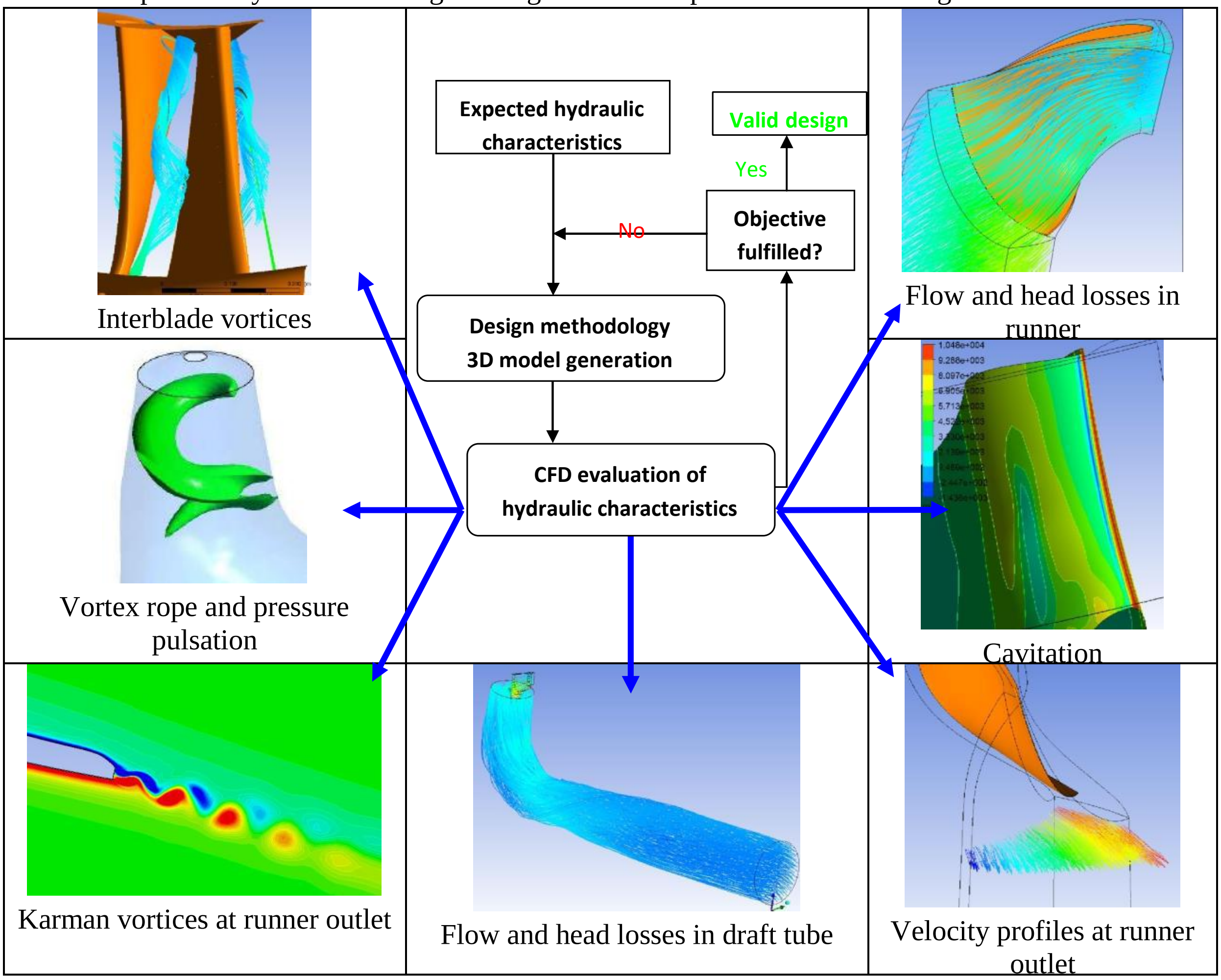


Figure 4 Optimisation Loop for runner and draft tube design

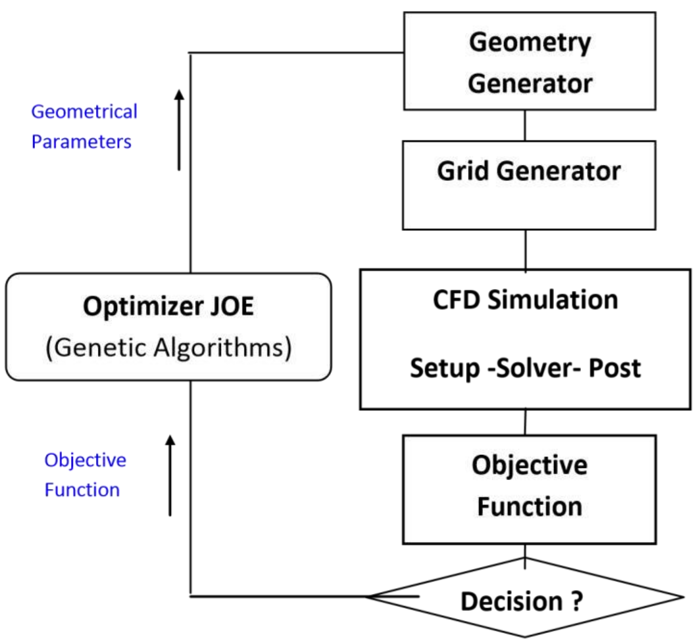
Taking into account these requirements, the hydraulic designer has to get high performances in the whole operating range while he has to address several topics : the head losses in the runner and in the draft tube, the inlet and the outlet cavitation limits for the runner, the velocity field at the outlet of the runner that drives the head losses and the pressure fluctuations in the draft tube. Each topic can be analysed and treated within an optimization loop as shown on the flowchart in Figure 4.

The optimization is often made manually : the shape of the runner is modified and adjusted to reach the different objectives that are the highest efficiency, no cavitation and the lowest dynamic loads. Nowadays, with the increasing computing capacity, this loop has been automated and becomes a very powerful tool that can help the hydraulic designer to define a runner blade. An example is presented in the following parts.

4.2. Runner design optimization

The principle of an optimization algorithm is to modify a vector of geometrical parameters called “individual” in order to increase the value of an objective function corresponding to this specific vector. Alstom has developed a toolbox called JOE (for Java Optimization Environment) using different optimalization algorithms, local methods as well as evolutionary ones. This optimizer manages and control all the software used in the optimal loop described in Figure 6: the geometry generator, the grid generator and the CFD software are coupled and are launched successively by the optimizer.

The geometry generator is an “in house” software that creates a surface parametric definition of the runner blade shape by using NURBS surfaces and a meridian channel. The skeleton of the blade is described by a camber-surface defined with 4x4 control-points as shown in Figure 5. Two thickness distribution surfaces that constitute the pressure and suction side surfaces, defined also by NURBS, are applied on the camber surface to create the blade shape (Figure 7).



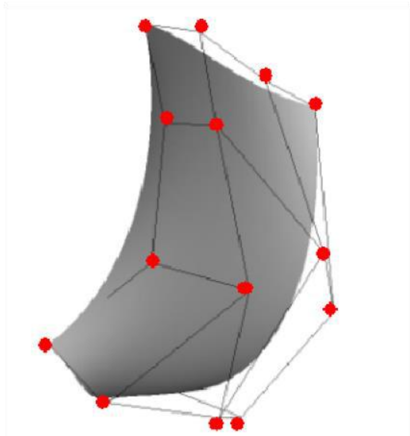


Figure 5 Example of camber surface

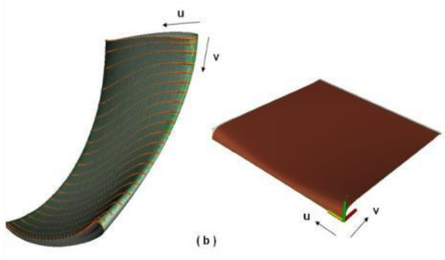


Figure 6 Design Optimization Scheme

Figure 7 Example of thickness distribution surfaces and blade

The grid generator is a robust “in house” software that generates automatically an hexaedric and Hstructured grid.

The CFD computation necessary for the design evaluation is performed using a Navier Stokes Solver running on a HPC Linux Cluster.

This optimization tool is more detailed in reference [3].

The optimal loop described above can be used to improve the performances and the dynamic behaviour of a large Francis turbine. In the multi–objectives optimalization example presented here, the runner and the draft tube performances were simultaneously optimized for the best efficiency and the high load condition. 24 geometrical parameters were used by the optimizer to generate the blade profile that delivers the flow velocity profile at the runner outlet in order to best fit the objectives.

The computational domain includes a runner passage and the draft tube. The computations are performed using a Navier-Stokes solver with a second order advection scheme and a SST turbulence model. Standard interface settings are used for the domain. Each computation is parallelized over 4 processors and the mesh sized has been limited to 300000 cells to ensure a reasonable computation time. Thus each computation needs less than one quarter of an hour to converge with a convergence criteria based on a maximum residual lower than 10^{-4}.

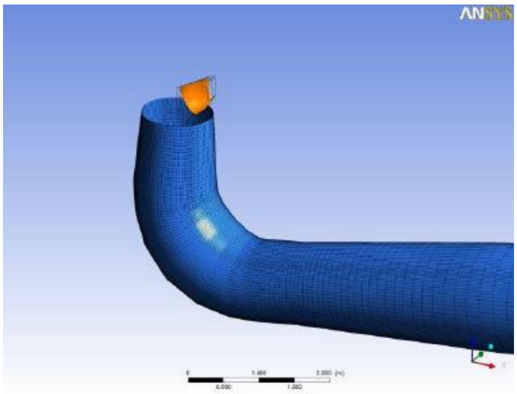


Figure 8 Domain and mesh used in the multi-objectives optimization

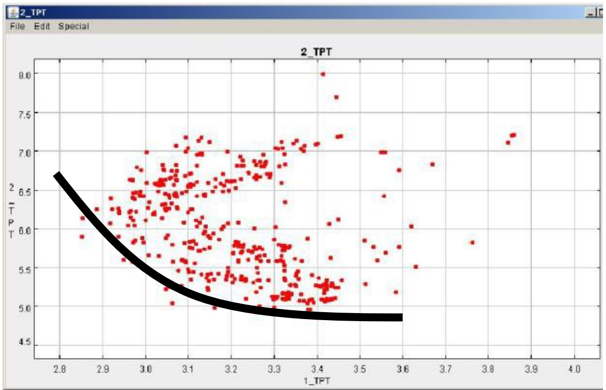


Figure 9 Pareto front obtained after the multi-objectives optimization – head losses at high discharge versus head losses at BEP

After a 1000 individuals computation, the optimization result is a Pareto front : the evolution of the head losses at high discharge is plotted as a function of the head losses at the Best Efficiency point on Figure 9.

Similarly, the optimizer loop is used to generate a blade profile that will best fit the most important characteristics expected for a project, such as performance, cavitation, pressure fluctuation or other dynamic phenomena.

If the optimal design tool is an appreciated help to find solutions when several non-independent objectives are expected, the experience of hydraulic engineers is still required to closely analyze the resulting geometries. To keep acceptable the time schedule, it is required to limit the criteria considered in the optimizer to the most relevant ones, and the hydraulic engineer knowledge is essential to insure that the retained blade shape will also met all the other expected characteristics of the project. At the end, all the hydraulic characteristics and phenomena are carefully studied and CFD calculated. It includes the cavitation phenomena, the flow dynamic phenomena that may generate pressure fluctuations such as the vortex rope and the inter-blade vortices. These dynamic phenomena are discussed here below.

4.3. Flow dynamic phenomena, pressure fluctuations in draft tube and stresses

As mentioned above, even if achieving the best possible efficiency is always a target, the flow dynamic phenomena that may result in dynamic loadings on the structure are, of course, key issues as they can affect the turbine lifetime.

In the runner and the draft tube, the main flow dynamic phenomena are :

- the rotor stator interaction presented above

- the inter-blade vortices
- the rotating rope
- the pulsation of the rotating rope

Thus, the dynamic behavior of the turbine is carefully studied. For instance, the Figure 10 and Figure 11 show that CFD helps to understand the generation of the vortex rope and the inter blade vortices.

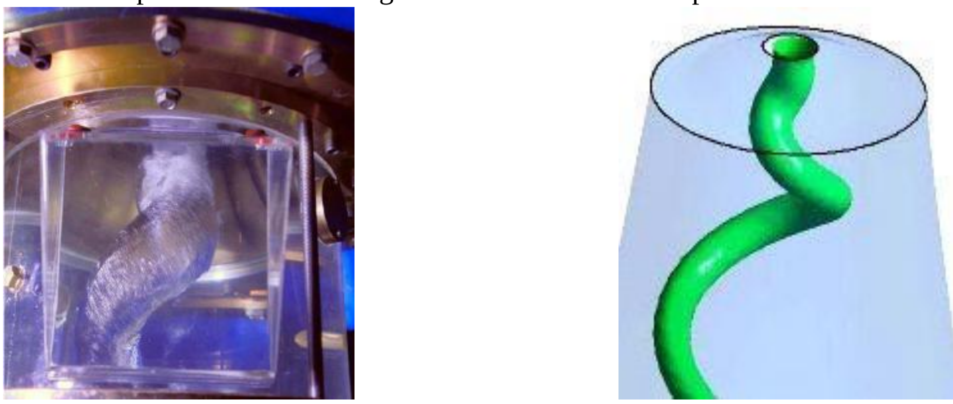


Figure 10 Vortex rope : measurement (left) and computation (right)

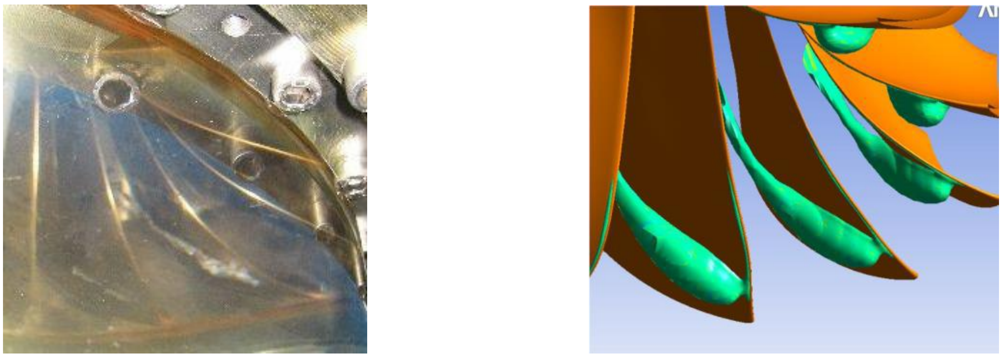


Figure 11 Inter-blade vortices: measurement (left) and computation (right)

It is important to determine which pressure fluctuation phenomena will have no impact on unit and which ones will impact either the unit integrity or the unit operation stability, to work on the appropriate phenomena.

For instance, pressure pulsations seen from the stationary water passages don’t necessarily correlate with high dynamic loading on the runner. A model runner equipped with stress gages (as shown on Figure 12) has been recently tested.

Figure 12 Francis runner blades equipped with strain gages

The pressure fluctuations in the draft tube cone and the runner stresses have been measured at a given head over a turbine discharge variation. On Figure 13, we observe that the pressure fluctuations peak measured in the draft tube around 0.7 Q/Q_{opt}, has no impact on the runner stress.

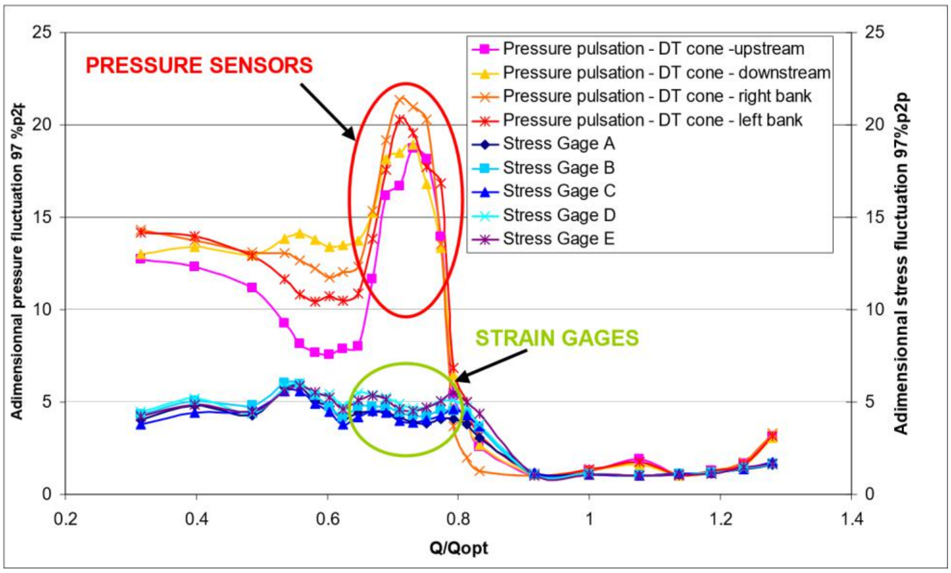


Figure 13 Pressures fluctuations in the draft tube cone and stresses on the blades as a function of Q/Qopt

This kind of measurements is helpful to show that some hydraulic phenomena, leading to pressure fluctuations in the draft tube, are not directly correlated to the runner dynamic loads and thus have no impact on its lifetime.

5. Conclusion

This paper presents part of Alstom Francis turbine design methodology combining intensive global and local optimization methods. These optimal methods are of great help to design all the turbine components (the upstream parts, the runner and the draft tube), and to meet all the expected turbine characteristics. The success of these optimization loops is widely associated to the growing computation capacity of the High-Performance Computing clusters. Such powerful clusters allow also to carry out a lot of unsteady computations and thus to take into account unsteady phenomena during the design stage.

In addition the runner is also model tested to further check and validate the design. These model tests are of course used to validate the performances, the cavitation limits and all the usual guarantees of a Francis turbine designed using CFD but, as they help also to better understand the physical phenomena occurring in a Francis turbine, they help to make a better use of CFD tools.

In any case, CFD and model tests have the same objectives : meet customers' expectations and increase turbine reliability.

References

[1] Mazzoui F, Segoufin C, Lowys P Y and Deniau J L 2006 Investigation of unsteadiness in hydraulic turbines *2rd IAHR Symposium on Hydraulic Machinery and Systems*, Yokohama, Japan

[2] Guillaume R, Deniau J L, Scolaro D and Colombet C 2012 Influence of the Rotor-Stator Interaction on the Dynamic Stresses of Francis Runners *26th IAHR Symposium on Hydraulic Machinery and Systems*, Beijing, China

[3] Flores E, Bazin D, Ferrando L and Mazzouji F 2007 Design optimization of a FRANCIS runner *Hydro 2007,* Granada, Spain