Proceedings of ENCIT 2012
14th Brazilian Congress of Thermal Sciences and Engineering
November 18-22, 2012, Rio de Janeiro, RJ, Brazil

OPTIMIZATION OF SUCTION TUBE BY USING PLAN OF EXPERIMENTS AND CFD TECHNIQUES

Tássio de Rezende, tassiode@uol.com.br
Edna Raimunda da Silva, ednaunifei@yahoo.com.br
Ramiro Gustavo Camacho Ramirez, ramirez@unifei.edu.br

Universidade Federal de Itajubá
Instituto de Engenharia Mecânica
Av BPS, 1303 Bairro Pinheirinho
37500-903 - Itajubá - MG
Brazil

Abstract. *In the developing of the design of flow machines and their components, one of the biggest challenges for the designer, in the incessant quest for increasing efficiency of turbomachinery, is to obtain an optimal geometry for the spiral case, runner, distributor and draft tube. In the first projects of hydraulic turbines, the experience of engineers and designers, along with numerous and costly tests with scale models, many of which are conducted on a "trial and error" basis, were the main design tools available. Part of this empirical knowledge accumulated was condensed into diagrams and guides (still used today), which provide good guidelines for pre-sizing for turbine draft tubes. Another part of this technology was retained with manufacturers as a legacy to future teams of engineers from the companies. Currently, the design of hydraulic turbines(rotor and draft tube) is made with increasingly intense use of computer programs for numerical calculation of the flow through the machine. These tools, like CFD (Computational Fluid Dynamics), are able to simulate accurately many important phenomena that occurs in the flow inside the turbine or the draft tub, helping the engineer understand these hydraulic passages details. Only after exhaustive investigations the reduced model is build to final tests at laboratories. The total time and costs spent on development of a new design is significantly reduced with this methodology, based in generation plane of experiments (DOE), heuristic optimization and metamodels construction. This paper aims propose a method of optimization for reduce the loss coefficient in the suction tube, from the construction of design of experiments(DOE)*

***Keywords**: Draft Tubes, Optimization , CFD, DOE*

1. INTRODUCTION

For hydropower companies, small improvements of hydraulic turbines are of interest due to economical and safety aspects. New demands on the regulation energy market make it also attractive to improve the turbines over a wide range of operating conditions. Furthermore, many of the turbines are old and need of rehabilitations and modifications. Hence, there is also a growing need of developing efficient design tools based on moderns methodologies in order to achieve efficient designs with economic returns for hydropower companies. Therefore, optimization techniques and Computational Fluid Dynamics CFD are integrated to improve the basic design.

An important component of a medium and low head reaction turbine is the hydraulic draft tube, since it is responsible for a large proportion of the total hydraulic losses in the system. However, it is also one of the most challenging parts to design, due to the interaction of many complex flow features, such as unsteadiness, turbulence, separation, streamline curvature, secondary flow, swirl, and vortex breakdown.

Traditionally, the form has been based on simplified analytic methods and rules of transposition between experimental tests of prototype and model (Gubin, 1973; Holmén, 1999).

Today, it is used computational fluid dynamics (CFD) to calculate the flow in the various components of hydraulic systems. Growth area is a very important factor due to its flexibility design and potential cost-effectiveness that can reach in saving energy by a adequate optimization concerning to its geometry. At the same time it is evident that the primary concern in industry is not to analyze and understand flow features, but whether improve the flow conditions. As a result, several CFD-based optimization frameworks have been suggested recently to improve the draft tube performance numerically (Eisinger & Ruprecht, 2001; Lindgren et al., 2002; Puente et al., 2003).

Time consuming CFD simulations of the draft tube flow are, however, impeding the optimization process, despite the steady advances in computer speed and capacity. Consequently, approximation methods, known as surrogate models, are widely used in similar optimization problems, to minimize the runtime dilemma. By this approach, it is possible to replace expensive CFD calculations with a surrogate model in the optimizations phase, providing a faster and more effective exploration of design and solution space. Thereby, also getting a better insight into the true relationship between objective functions and design variables.

The choice of a suitable surrogate model is moreover important because it should represent the physical model obtained from CFD calculation more accurately. Surrogate models based on radial basis functions has shown satisfactory results when were evaluated with function test. Silva has applied interpolation functions of radial basis (RBF) and polynomials in the design of hydraulic turbine rotors and turbomachinery cascades.

2 DRAFT TUBE

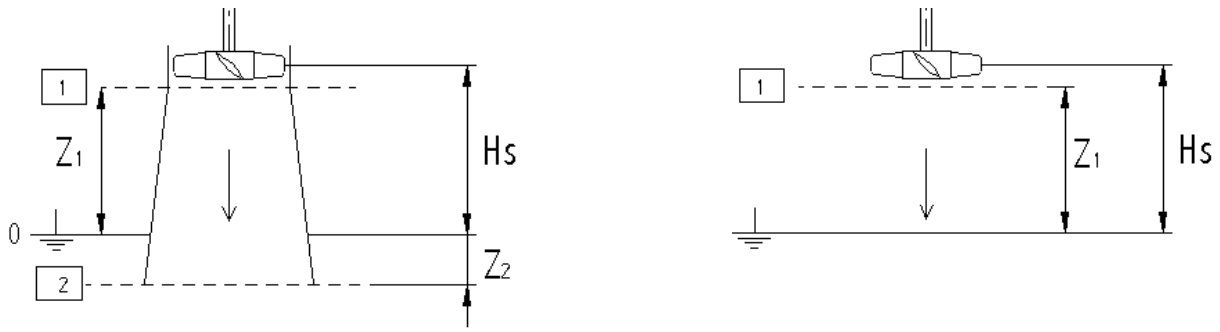
The principle of a draft tube can be outlined by aid of Bernoulli's equation between section 1 and 2 (inlet and outlet, respectively) in Figure 1:

$$\frac{p_1}{\rho g} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\rho g} - z_1 + \frac{V_1^2}{2g} + h_f \tag{1}$$

Here p is the absolute pressure, z the height, V the medium velocity and h_f the hydraulic losses in the draft tube. The absolute pressure p at section 2 can furthermore be expressed as $p_2 = z_2\rho g + p_{atm}$, where p_{atm} is the atmospheric pressure. Assuming that the turbine installation height H_s is approximately equal to z_1, Eq. (1) reduces to:

$$\frac{P_1}{\rho g} = \frac{P_{atm}}{\rho g} - \left(H_s + \left(\frac{\alpha_1 V_1^2}{2g} - \frac{\alpha_2 V_2^2}{2g} - h_f\right)\right). \tag{2}$$

An interpretation of Eq. (2) is that the draft tube generates a low pressure region underneath the runner, which can be utilized by the turbine. This lower pressure, consists of two terms; static fall of pressure and dynamic fall of pressure, Hs and $(V_1^2 - V_2^2/2g)$-h_f, respectively. The former part is



(a) **(b)**

Figure 1 - Hydraulic principle of draft tubes;
(a) with; (b) without.

independent of the discharge while the latter part generally increases with the flow rate. This second precaution is easily accomplished by increasing the diffuser wall angle and/or by enlarging the draft tube length. In both cases, however, the hydraulic losses will become larger. An efficient draft tube has therefore an optimal diffuser angle and length for which the pressure reduction below the runner will be maximal (Gubin, 1973).

The efficiency of a draft tube (or a diffuser) is generally described by four performance metrics. These are the actual pressure recovery Cp, the ideal pressure recovery C_{pi}, the draft tube efficiency η_{cp}, and the loss factor, respectively. They are furthermore usually defined as;

$$C_p = \frac{P_2 - P_1}{\frac{V_1^2}{2}} \tag{3}$$

$$C_{pi} = \left(\frac{V_2}{V_1}\right)^2 = \left(\frac{A_1}{A_2}\right)^2, \tag{4}$$

$$\eta_{cp} = \frac{C_p}{C_{pi}}, \text{ and} \tag{5}$$

$$\zeta = C_{pi} - C_p = 1 - C_p - \left(\frac{A_1}{A_2}\right)^2, \qquad \text{respectively,} \tag{6}$$

where A is the cross-section area. Depending if maximum pressure recovery Eq.(3) or maximum draft tube effectiveness Eq. (5) is wanted, different optimal diffuser wall angles will be found (Kline et al., 1959).

Clearly, there are more benefits to using the draft tube, because a large increase of pressure energy is carried out, where the lower value of output velocity (v_2), represent a high recovery of energy. However, this velocity is restricted by the maximum possible variation of pressure between input and output. This variation of pressure is proportional to the growth rate of the area of the draft tube. The high growth rates can present problems of boundary layer separation and vortex emissions, resulting in zones of recirculation, which makes this diffuser tube is inefficient. Therefore, the objective of this study aims to determine optimal geometry that can improve the overall efficiency of the suction tube.

3. METHODOLOGY

3.1 Optimization – CRSA (Controlled Random Search Algorithm)

To determine the plan experiences CRSA program was used, where the initial population is generated based on randomized criteria. The program allows to generate the initial population and to the search for objective function chosen. Due to the high computational time, only was analyzed the experiments plane.

A Controlled Random Search Algorithm (CRSA) is an iterative, stochastic, population-set based algorithm, capable of finding the global minimum of real functions efficiently. Promotes the substitution of the worst point of the population by a better one at each iteration. CRSA is an algorithm based on the generation of a population of initial points P of N randomly generated points on *S*, that, following an iteration process, converges into a global minimum by purely heuristic procedures. The population size N is maintained along the optimization process. CRSA substitutes an only point of the population (your worst point, h) for a better point l in each iteration (it is., a test point l so that *f*(l) < *f*(h)). Your implementation is direct and was introduced by Manzanares-Filho et al. (2005).

3.2. Geometry Parameters

The draft tube was parameterized using circular sections according to the following parameters (See Figure 2) :

- Output Angle (α)
- Inlet diameter (*d*)
- Radius of curvature (*r*)
- Outlet length (*l*)
- Growth area factor (*f*)
  This factor multiplies the value of the area in the previous section for determining the diameter of the new section. This way, it can be determined the percentage of linear growth areas along the circular sections of the draft tube.

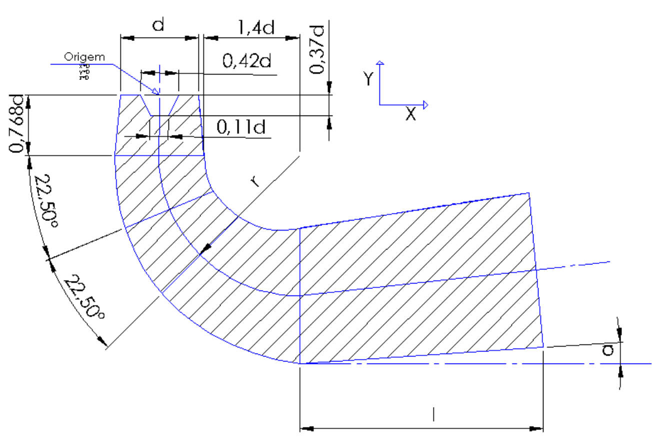


Figure 2 – Cut view of the frontal plane of the parameterized draft tube

3.2 Boundary Conditions

Inlet.

Based on the work of Ruprecht (2005) were inserted into user-defined functions (UDF) in order to introduce the boundary conditions at the inlet tube, i.e. components of velocity axial, radial and tangential as well as the profiles turbulent kinetic energy (k) and dissipation (ε) (See Figures 3-8).

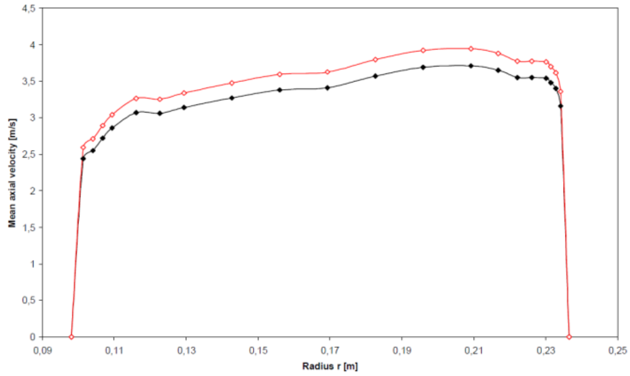


Figure 3. – Axial velocity vs. radius inlet

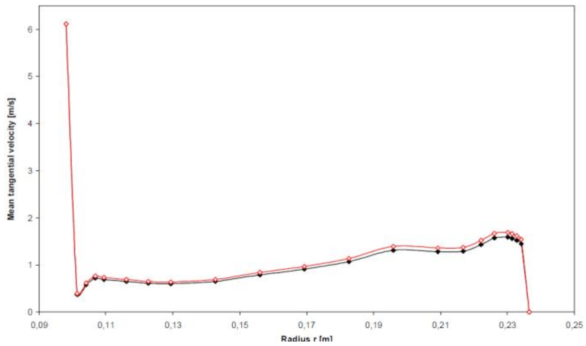


Figure 4 –Tangential velocity vs. radius inlet

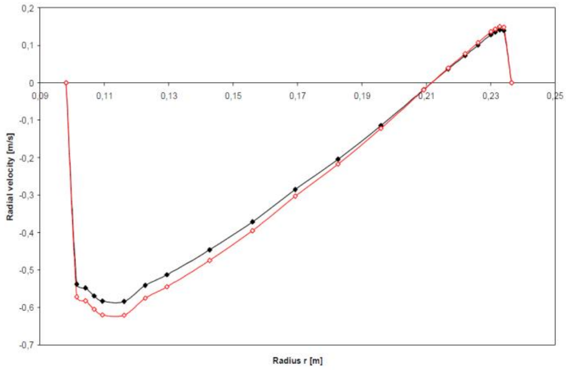


Figure 5 – Radial velocity vs. radius inlet

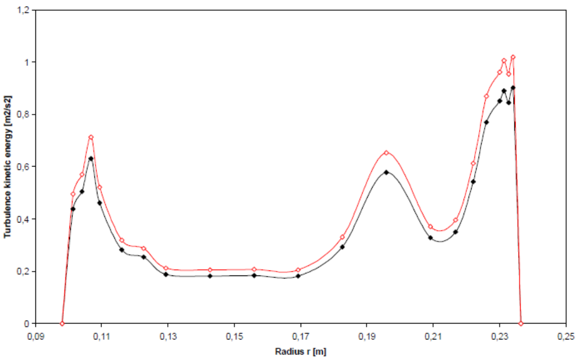


Figure 6 – Turbulent Energy *k* vs. radius inlet

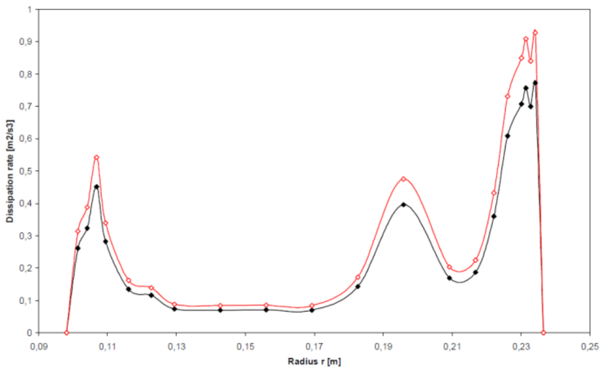


Figure 7 – Turbulence dissipation ε vs. radius inlet

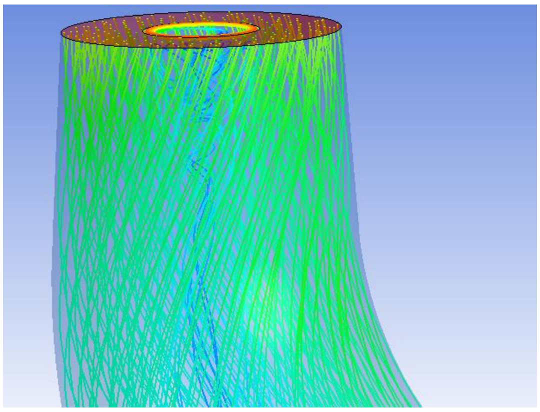


Figure 8 – Boundary conditions on the input surface, obtained through of UDF and swirl formation

All these data were proceeded and exported in following table:

Table 1 – Boundary conditions at the inlet

	Radius (*r*)	Axial Velocity	Tangential Velocity	Radial velocity	*k* Turbulence	*ε* Dissipation
Unities	*M*	*m/s*	*m/s*	*m/s*	m^2/s^2	m^2/s^3
0	0,090	0,000	0,000	0,000	0,000	0,000
1	0,098	0,000	6,171	0,000	0,000	0,000
2	0,101	2,430	0,369	-0,546	0,442	0,262
3	0,104	2,535	0,579	-0,556	0,510	0,323
4	0,107	2,706	0,722	-0,578	0,636	0,454
5	0,110	2,844	0,690	-0,592	0,464	0,282
6	0,116	3,053	0,648	-0,592	0,283	0,135
7	0,123	3,043	0,613	-0,548	0,255	0,117
8	0,130	3,121	0,597	-0,520	0,189	0,073
9	0,143	3,253	0,648	-0,453	0,182	0,069
10	0,156	3,362	0,793	-0,378	0,185	0,071
11	0,170	3,389	0,919	-0,289	0,178	0,071
12	0,183	3,552	1,077	-0,208	0,295	0,143
13	0,196	3,672	1,319	-0,116	0,581	0,397
14	0,210	3,691	1,292	-0,019	0,330	0,170
15	0,217	3,632	1,292	0,037	0,352	0,187
16	0,223	3,530	1,436	0,073	0,547	0,362
17	0,227	3,530	1,590	0,101	0,777	0,611
18	0,231	3,518	1,607	0,129	0,859	0,711
19	0,232	3,455	1,573	0,136	0,897	0,759
20	0,234	3,380	1,532	0,143	0,852	0,701
21	0,235	3,142	1,462	0,140	0,910	0,776
22	0,237	0,000	0,000	0,000	0,000	0,000

Based on these data it is possible to define interpolation functions denominated UDF (User Define Functions) to be interpreted by FLUENT® software.

-Boundary conditions in the cube

The cube of rotor Kaplan is regarded as an animated wall. It has a rotational movement around the central shaft at the entry with the value of 595 rpm

-Wall boundary conditions
The tube walls are only considered to be stationary.

-Boundary conditions in the output
In the output of the draft tube condition was imposed on "Outflow" of FLUENT ®. Such a condition is used when there are not details about the fields of pressure and velocities in a sector of the flow, and it is known only that the whole entry mass flow in the system flows there.

3.3 Simulation

With a determined geometry, and boundary conditions was performed the first simulation of the model. In the simulation, was used, the *k-ε* turbulence model, 2000 iterations were performed until the convergence of calculation. The time taken for each simulation was approximately 5 hours on a personal computer with 8 cores and 16 Gb RAM. The Figure 9 and 10 shows the streamlines at the draft tube and pressures contours.

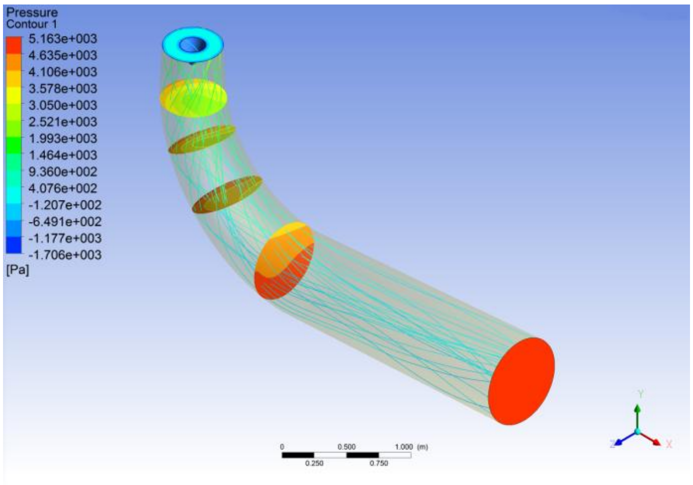


Figure 9 – Streamlines at the draft tube

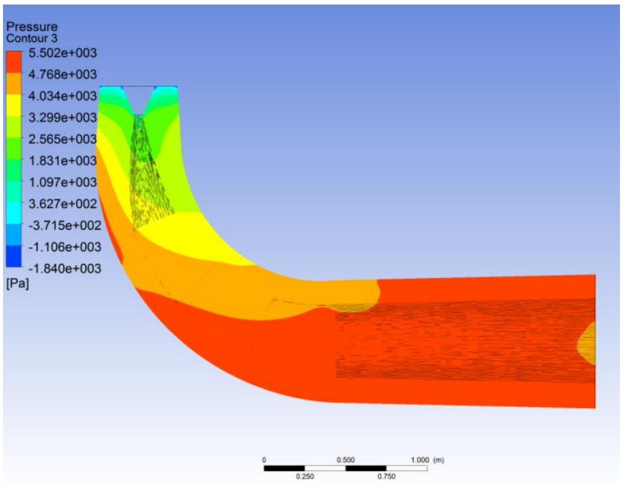


Figure 10 – Pressures contours at the draft tube

Planning of experiments

In order to generate the plane of experiments were selected as design variables radius of curvature and the exit angle keeping other variables fixed, and then; $N_{pop}=(N_{niv})^{N\text{-}var}=25$; $N_{niv}=5$; $N_{-var}=2$

4. CRITERIA OF SELECTIONS FOR CALCULATING LOSSES

Chosen the variables to investigate, it is necessary to determine criteria for the assessment of the performance of each experiment, in order to infer a pattern and an optimum point to be used as a guideline for the design of a suction tube. Therefore, the evaluation used two criteria.

The first criterion is the average pressure coefficient (C_{pm}), which is a relationship between average static pressure difference (at the inlet and outlet of the tube) and dynamics medium pressure in the tube input. Suction tube design was based on the concept of a diffuser, doing increases in the area progressively rises the static pressure and decreases dynamic pressure (velocity), the previous mentioned situation is proportional to the pressure coefficient.

$$Cp_m = \frac{\frac{1}{A_{out}}\int_{out} P_{static} d\dot{A} - \frac{1}{A_{in}}\int_{in} P_{static} d\dot{A}}{\frac{1}{A_{in}}\int_{in} P_{dynamic} d\dot{A}} \tag{7}$$

The second criterion is the loss coefficient (ζ), this lists the amount of input energy given by the total pressure at the entrance and the total pressure in the output with the dynamic pressure on input (in average terms). Therefore allows to assess the amount of energy was lost through phenomena such as friction, vortex formation and detachment of the boundary layer.

$$\zeta = \frac{\frac{1}{A_{out}}\int_{in} P_{total} d\dot{A} - \frac{1}{A_{in}}\int_{out} P_{total} d\dot{A}}{\frac{1}{A_{in}}\int_{in} P_{dynamic} d\dot{A}} \tag{8}$$

All necessary variables were calculated by CFD - Post after the end of each simulation. For the calculation, the pressures and velocities have been given by *area average* and *mass flow average*. The first executes a average weighted of all variables only on the second area and takes into account the mass flow of water through the sections. The second provides more reliable from the standpoint of turbulent flows associated with regions of recirculation, since there may be dead zones (recirculation) flow which results in a diffuse area.

In the Table 3 shows the results of the 25 experiments or simulations 25 of approximately 2 hours and 30 minutes, a total of 62.5 hours for processing. Were considered as fixed variables diameter at the entrance, the growth factor of each section equal to 20%, and the tube length l = 2500 mm. In table 3 are reported the results of the loss coefficient and the pressure coefficient calculated between the input and output surfaces of the suction tube, using the equations 7 and 8.

Table 3. Results of the experiments

Exp. [-]	**α** [º]	**R** [mm]	*Area average* ζ loss coefficient	C_{pm}	*Mass average* ζ loss coefficient	C_{pm}
1	2	700	0.0767	0.8443	0.0929	0.8261
2	2	900	0.0734	0.8277	0.0920	0.8081
3	2	1100	0.0645	0.8374	0.0808	0.8187
4	2	1300	0.0793	0.8376	0.0948	0.8197
5	2	1500	0.0824	0.8355	0.0997	0.8173
6	4	700	0.0796	0.8362	0.0965	0.8185
7	4	900	0.0774	0.8271	0.0944	0.8078
8	4	1100	0.0907	0.8289	0.0898	0.8099
9	4	1300	0.0809	0.8402	0.1122	0.8225
10	4	1500	0.0911	0.8262	0.1068	0.8080
11	6	700	0.0733	0.7535	0.0905	0.8339
12	6	900	0.0906	0.8304	0.0911	0.8110
13	6	1100	0.0734	0.8339	0.0886	0.8157
14	6	1300	0.0778	0.8344	0.0948	0.8168
15	6	1500	0.0869	0.8292	0.1035	0.8109
16	8	700	0.0815	0.8368	0.0978	0.8143
17	8	900	0.0690	0.8359	0.0873	0.8170
18	8	1100	0.0747	0.8331	0.0894	0.8150
19	8	1300	0.0953	0.8244	0.1100	0.8070
20	8	1500	0.1147	0.8057	0.1225	0.7893
21	10	700	0.1248	0.7887	0.1239	0.7732
22	10	900	0.1209	0.7957	0.1233	0.7797
23	10	1100	0.1224	0.7942	0.1244	0.7778
24	10	1300	0.0874	0.8307	0.1007	0.8139
25	10	1500	0.1164	0.8011	0.1212	0.7845

5 RESULTS.

Based on the results shown in Table 3, it is possible to obtain the contours of the loss coefficient and the pressure coefficient in function of radius of curvature and angle alpha, offering excellent geometries, with minimal losses. There are, in both graphs, that in regions near the center (Fig. 10) are the best conditions for minimum loss coefficient values. However, in regions with small curvatures and angles alpha largest, may be regarded unstable parts of high losses.

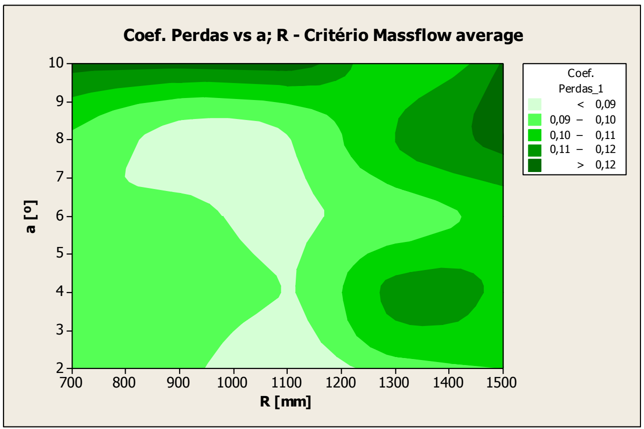


Figure 11. Contours of loss coefficient

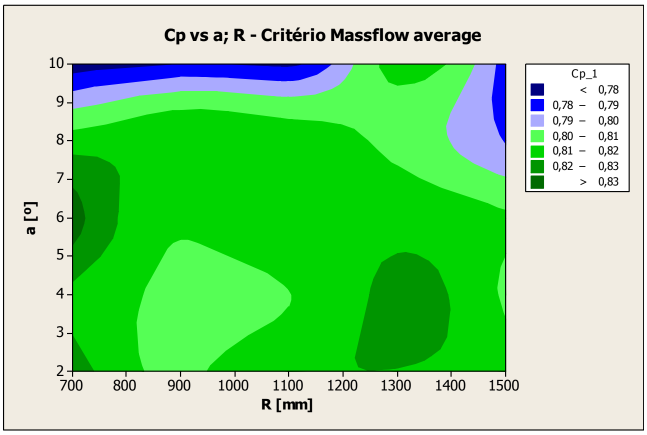


Figure 12. Contours of pressure coefficient

6. CONCLUSIONS

Based on preliminary results: geometry parameterization, automatic mesh generation (hexaedrical), analyzes of local and global variables of the flow field, can be integrated with optimization algorithms for generated a plane of experiments aiming to find the optimal geometry so as to reduce losses.

In this paper, only two variables were considered for the project, as the radius of curvature and angle of inclination at the exit of the tube. However other geometric variables can be considered in order to achieve optimal solutions. It is important that besides the calculation of the doe, the project of the suction tube must be optimized based on genetic algorithms and metamodels constructions. Future studies are being developed in the group LHV (Virtual Hydrodynamics Laboratory) of UNIFEI, with several applications in turbomachinery

7. ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support of FAPEMIG, the state of Minas Gerais - Brazil and UNIFEI.

8. REFERENCES

Manzanares-Filho N., 2009. Silva E. R., Lima M. G., Ramirez C. R. G., Blade Cascade Optimum Design Using a Stochastic Search Algorithm and a Metamodeling Approach. 8th World Congress on Structural and Multidisciplinary Optimization, Lisbon, Portugal, June 1 - 5.

Ruprecht A., 2005. Numerische Stromungssimulation am Beispiel hydraulischer Stromungsmaschinen, Habilitation thesis, University of Stuttgart - Turbine-99 Workshop III.

Gubin, M.F., 1973. "Draft tubes of Hydro-Electric Stations", Amerind publishing Co, New Dehli,

Holmén, E., 1996 . "Draft Tubes – from a Farmer's Trick to Efficient Water Turbines", Proc. of Turbine-99 Workshop on Draft Tube Flow, Technical Report 2000:11, Luleå University of Technology, Sweden.

Eisinger, R., Ruprecht, A., 2001. "Automatic Shape Optimization of Hydro Turbine Components based on CFD", TASK Quarterly, Vol 6, pp 101-111.

Lindgren, M., Marjavaara, B.D., Lundström, T.S., 2002. "Automatic Design of Hydropower Flows: The Draft Tube" American Society of Mechanical Engineers, Pressure Vessels and Piping Division (Publication) PVP, Vol 448, No 1, pp 299-307

Puente, L.R., Reggio, M., Guibault, F., 2003. "Automatic shape optimization of a hydraulic turbine draft tube" Proc. of the 11th Annual Conference of the CFD Society, Canada, May 28-30.

Kline, S. J., Abbott, D. E., Fox, R. W., 1959. "Optimum Design of Straight Walled Diffuser", Journal of Basic Engineering, Vol 81, pp 321-329.

9. RESPONSIBILITY NOTICE

The author are the only responsible for the printed material included in this paper.