

# Hydraulic analysis and optimization design in Guri rehabilitation project

H Cheng<sup>1,2</sup>, L J Zhou<sup>2</sup>, L Gong<sup>1</sup>, Z N Wang<sup>1</sup>, Q Wen<sup>1</sup>, Y Z Zhao<sup>1</sup> and Y L Wang<sup>1</sup>

<sup>1</sup> Dongfang Electrical Machinery Co., Ltd., No. 188 Huang He West Road, Deyang, Sichuan Province, P. R. China 618000

<sup>2</sup> College of Water Resources and Civil Engineering, China Agricultural University, Beijing, P. R. China 100083

E-mail: lodgecheng@163.com

**Abstract.** Recently Dongfang was awarded the contract for rehabilitation of 6 units in Guri power plant, the biggest hydro power project in Venezuela. The rehabilitation includes, but not limited to, the extension of output capacity by about 50% and enhancement of efficiency level. To achieve the targets the runner and the guide vanes will be replaced by the newly optimized designs. In addition, the out-of-date stay vanes with straight plate shape will be modified into proper profiles after considering the application feasibility in field. The runner and vane profiles were optimized by using state-of-the-art flow simulation techniques. And the hydraulic performances were confirmed by the following model tests. This paper describes the flow analysis during the optimization procedure and the comparison between various technical concepts.

## 1 Introduction

The Guri hydropower station, located on the Rio Caroni River, is the largest hydropower project in Venezuela. The total installed capacity of Guri hydropower station, which was constructed in two stages, is 10300MW. There were 10 units erected in the first stage with rated capacity of 266MW, and another 10 units were erected in the second stage with rated capacity of 600MW [1]. Because of the great change in operating conditions, the hydraulic turbines of Guri suffered from serious safe problems, such as vibration and cavitation. The rehabilitation of 10 units has been implemented in Guri, whose turbine is one of the biggest capacity machines to be rehabilitated all over the world.

Dongfang Electrical Machinery Co., Ltd (DFEM) got the contract for rehabilitation of 1-6# units in Guri in August 2015, and this project include that DFEM should enhance the maximal capacity of 1-3# units from 220MW to 300MW and 4-6# units from 270MW to 400MW. According to the contract, the preliminary model test report should be provided within eight months after signing, which is a very short time for design and development of the hydraulic turbine.

## 2 Operating parameters and characteristics

The parameter  $H_{\max}/H_{\min}$  is equal to 1.3, so the variation in power plant operation head is larger, which is disadvantageous to the stable turbine operation [2]. These parameters ( $H_{\min}/H_r=0.8$ ,  $H_{\max}/H_r=1.04$ ) prove that the station should have a higher design head, which is better to enhance stability at high head. Table 1 shows the main hydraulic parameters.



Based on statistics for turbines operating over 100m head, the incipient cavitation coefficient  $\sigma_i$  is about from 0.11 to 0.13, and the critical cavitation coefficient  $\sigma_1$  is from 0.06 to 0.08. Guri project's plant sigma  $\sigma_p$  at rated point is about 0.14 due to existed excavation depth. According to the experiential incipient safety margin  $K_i=1.1$  and critical safety margin  $K_1=1.6$  [3], it is a challenge for hydraulic development of Guri turbine to fulfill the specification of the cavitation performance.

With the rated speed 138.46 and 128.57 rpm, we could calculate that rated point specific speed  $n_s$  are 163 and 175 m.kw, with the speed coefficient 1904 and 2043. Compared with the statistics of the main manufacturers in the world [4], this specific speed is close to the average statistical value, which is favorable to the economical concerns and operating stability of the station [5].

**Table 1.** Hydropower station parameters

Item	1-3# units		4-6# units	
	original	rehabilitated	original	rehabilitated
Max. head, $H_{\max}$ (m)	115	142	128	142
Rated head, $H_r$ (m)	92	136	92	136
Min. head, $H_{\min}$ (m)	70	109	70	109
Rated speed $n_r$ (rpm)	128.57	138.46	120	128.57
Rated discharge $Q_r$ ( $m^3/s$ )	220	215	272	294
Max. output $P_{\max}$ (MW)	220	300	270	400
Setting elevation $\Delta_s$ (m)	120	120	120	120
Runner nominal diameter $D_1$ (m)	5.2	5.26	5.57	5.85
Suction head $H_s$ (m)	\	-8	\	-8

### 3 Hydraulic development

The model turbine development for Guri was based on a combination of selection and design. Firstly, a suitable model runner was selected from the database according to the specified hydraulic parameters and operating range. Secondly, the selected hydraulic model was fine tuned based on the actual civil sizes of the power station, especially the guide vanes and runner. The turbine model was tested internally by DFEM, and the hydraulic performance and stability of the design was evaluated. The model was then adjusted, if necessary, to improve the performance. Finally, when the specified performance had been proven internally, the model test was witnessed and approved by the Client.

**Table 2.** The main parameters of model and prototype

Item	1-3# unit		4-6# unit	
	Model	Prototype	Model	Prototype
Runner nominal diameter $D_1$ (mm)	377.58	5260	377.28	5850
Blade number F	15	15	17	17
Stay vane number	24	24	24	24
Guide vane shape	Negative	Negative	Negative	Negative
Height of Guide vane (mm)	85	1184	85	1318
Guide vane number Z	24	24	24	24
Diameter of Guide vane circle (mm)	432.1	6020	432.1	6700

DFEM has supplied turbines for a number of large and medium-sized hydro power plants with the head over 100m, such as the Three Gorges, Longyangxia, Liji Xia, Jinganqiao plants, and all of them were tested. The development of the turbines for Guri plant is based on a large scale of technical database and experience. Table 2 shows the size of the model turbine and the expected hydraulic parameters of the prototype.

### 4 CFD numerical analysis

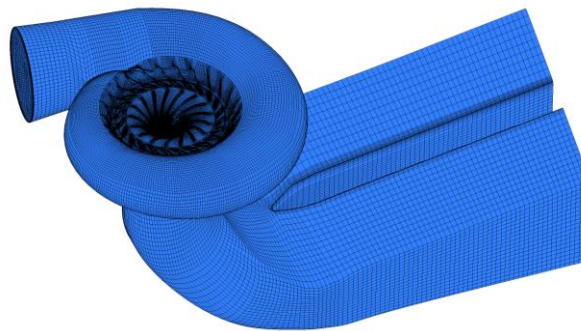
Computational Fluid Dynamic (CFD) is the mainly used mathematical method for hydraulic turbine's

optimization [6]. It is now quite a mature and reliable technology [7], whose calculation speed and accuracy can meet the engineering requirements. There are two steps in the optimization process. The first is to develop a preliminary design of single component, based on experience and specific parameters, and then to subject this preliminary design to the optimization loop. The second step is to calculate the turbine with multi domains or even with the whole flow passage, which is intensively introduced in this paper.

3D steady state flow simulation is carried out for the model turbine. The commercial software ANSYS CFX V13.0 is employed and the Shear Stress Transport (SST) turbulence model is used for the simulation. Various operating points, e.g. part load, optimum, rated load, and over load points, are selected for comparison. The simulation results have been compared with the model test results to validate the numerical accuracy and reliability. Due to the same methods of analyzing Guri project 1-6# units, we would take the 3-6#units for an example below.

#### 4.1 Simulation setup

The simulation domain is the full passage of a Francis turbine, including spiral casing, stay vanes, guide vanes, runner, and draft tube. The software ANSYS ICEM CFD V13.0 and ANSYS Turbo Grid V13.0 are used to create the mesh of domain. In this paper all the components are meshed with hexahedron element type, as shown in figure 1 , which is more adaptive to ensure the precision [8].



**Figure 1.** Simulation domain and mesh of the model turbine

The total pressure representing 30m head is given at spiral casing's inlet as inlet boundary condition. For the outlet boundary condition zero average static pressure is given at draft tube outlet [9]. All wall surfaces are treated as no-slip wall to consider the friction loss. The domain of runner is connected with other components of the passage by interface of Frozen Rotor. RMS residual type is chosen as convergence criteria, which is 1e-04 in most condition.

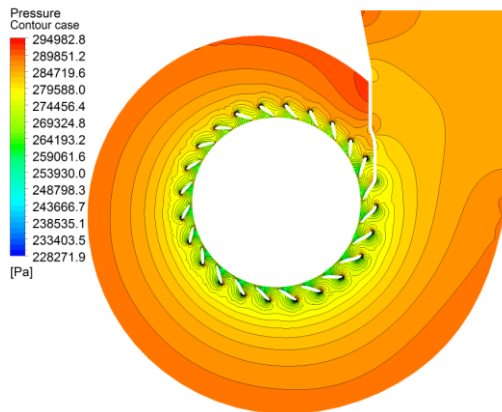
**Table 3.** Number of mesh

Item	Spiral casing	Stay vanes	Guide vanes	Runner	Draft tube
Elements number	282684	528012	768384	1694900	255433
Nodes number	298158	586124	834048	1794690	266180
Yplus average value	<60	<60	<50	<70	<40

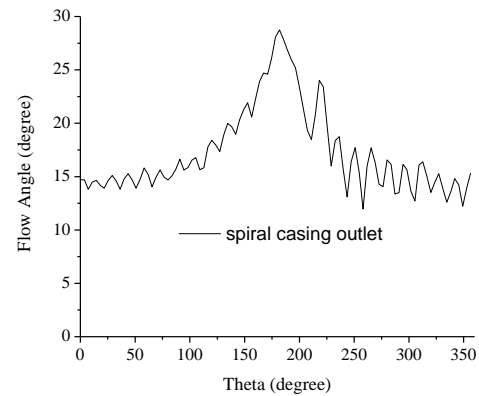
#### 4.2 Spiral casing

It's necessary to analyze the flow pattern of the original spiral casing, combined with other parts of the turbine. If the spiral casing passage could provide suitable flow condition, a uniform velocity and required flow angle at the stay vane's inlet can be achieved as well as the minimized hydraulic loss.

Figure2 shows that the pressure distribution is smooth and uniform in spiral casing at optimum condition. The maximum variation of flow angle at the outlet of spiral casing, shown in figure3, is just 15 degrees, which is acceptable to the downstream component of the turbine. Therefore, we could decide that original spiral casing still has good performance, and it's helpful to carry out the task of optimization design.

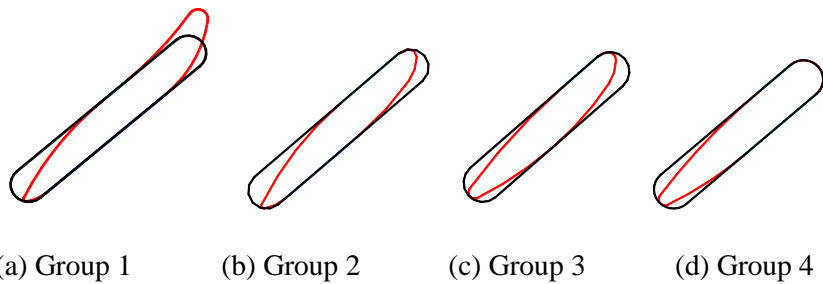


**Figure 2.** Pressure contour on the mid-span level of spiral casing in optimum condition

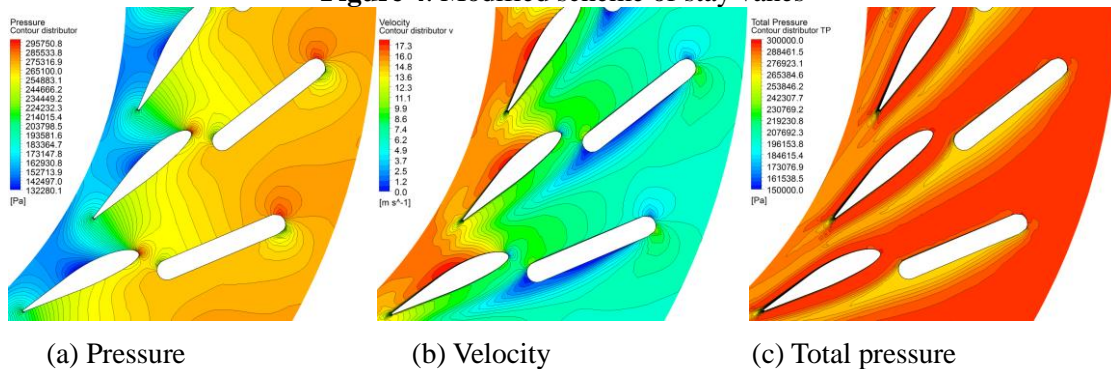


**Figure 3.** Flow angle at the outlet of spiral casing in optimum condition

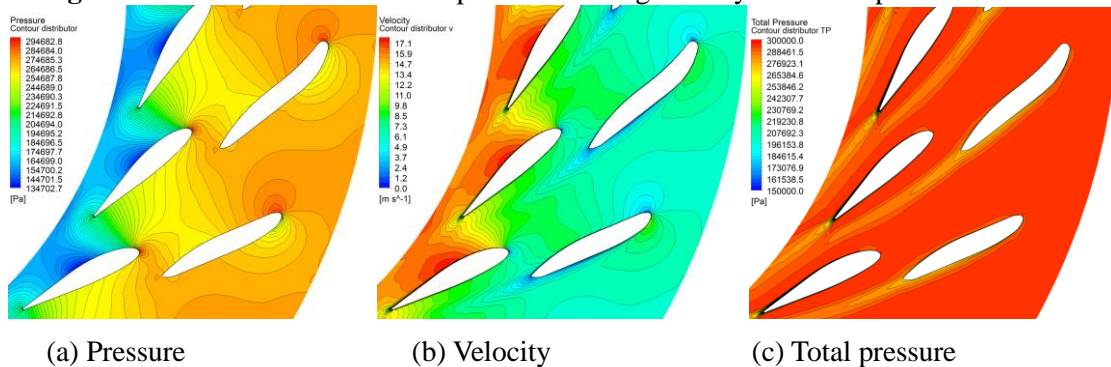
#### 4.3 Distributor



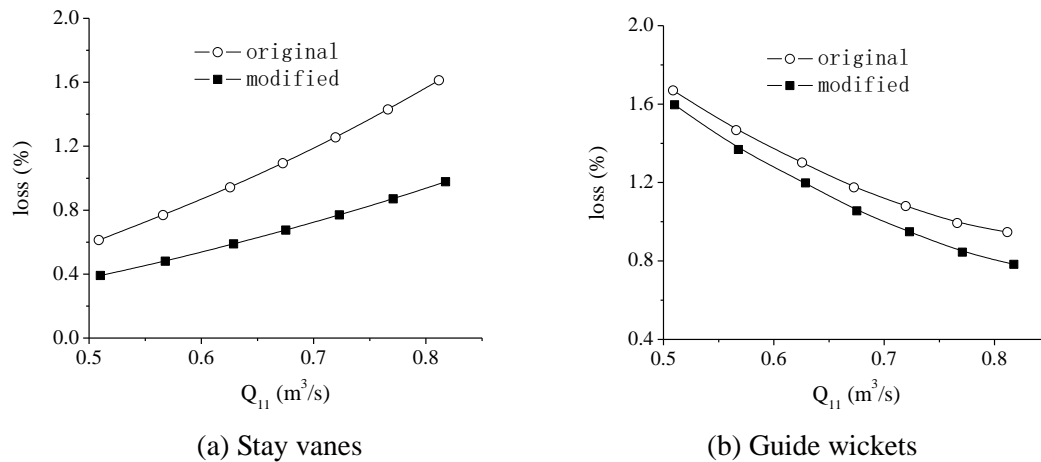
**Figure 4.** Modified scheme of stay vanes



**Figure 5.** The contour on the mid-span level of original stay vanes in optimum condition



**Figure 6.** The contour on the mid-span level of modified stay vanes in optimum condition



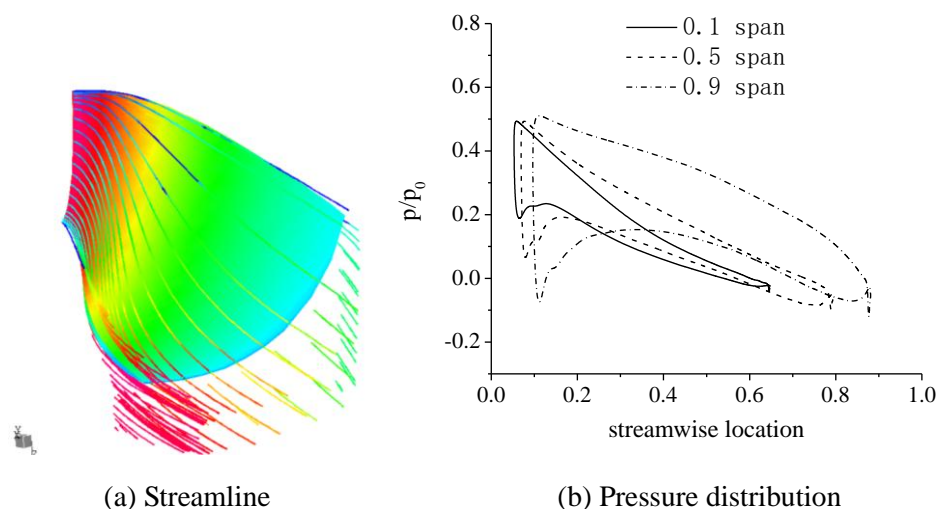
**Figure 7.** Comparison of the original and modified distributor hydraulic loss at design head

The function of distributor is to control the flow towards the runner. The main aim in the design of the complete distributor is to optimize the shape of the hydraulic components and establish the relationship between the stay vanes and guide wickets, thereby enhancing the overall turbine performance at all operating conditions in the specified head range.

However, through simulating the flow pattern in distributor, as shown in figure 5, we could find that there are negative attack angle in the leading edge and flow separation in the tailing edge of most stay vanes. Those unsuitable flow phenomena greatly increase the hydraulic loss, therefore, DFEM determine to modify the old profile of stay vanes. Considering the mechanical intensity and project's cost, the final scheme is illustrated in figure 4.

The final results show that, at optimum condition, the pressure and velocity distribution, as shown in figure 6, in each vanes are more uniform, and the dissipation of total pressure is significantly less than original scheme. Meanwhile, due to DFEM's excellent database of guide wickets, the flow pattern at the guide vane's outlet can satisfy the flow requirements of the runner. The distributor's hydraulic loss decreases by about 0.4% at stay vanes and 0.1% at guide wickets in different operated conditions, as shown in figure 7.

#### 4.4 Runner

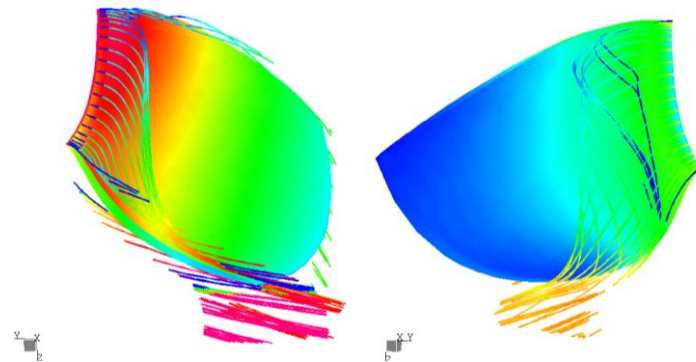


**Figure 8.** Flow pattern of runner blade in the optimum condition

During turbine runner development, most of the works focus on the flow pattern together with the



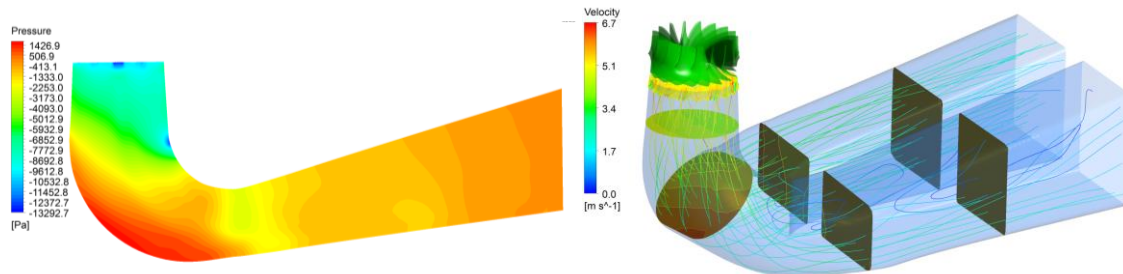
guide vanes, which could externally simulate the influence of the out flow of guide vanes on the runner [10]. The CFD calculation includes these key operating points: the optimum, high load at low head, part load at high head condition. We could analyze the streamline and pressure distribution to judge the optimum point, as shown in figure 8. The aim of calculating the part load condition at high head is to check the inter blade vortex incipience. Meanwhile, calculating the high load condition at low head is to check the flow separation on the pressure side of runner blade due to negative incidence at the blade leading edge. Figure 9 verifies that the blade channel vortex and flow separation are basically acceptable at these operated points.



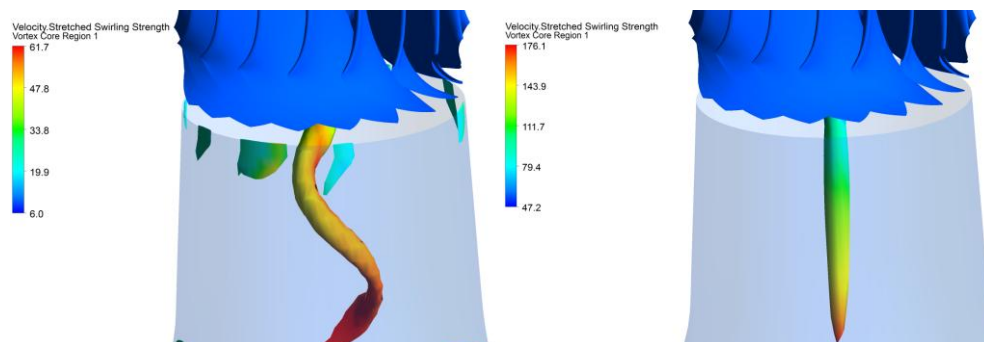
(a) High load at low head (b) Part load at high head

**Figure 9.** Flow pattern of runner in other condition

#### 4.5 Draft tube



**Figure 10.** Pressure contour and streamlines in the draft tube at optimum point



(a) Partial load

(b) High load

**Figure 11.** Rope vortex in the draft tube

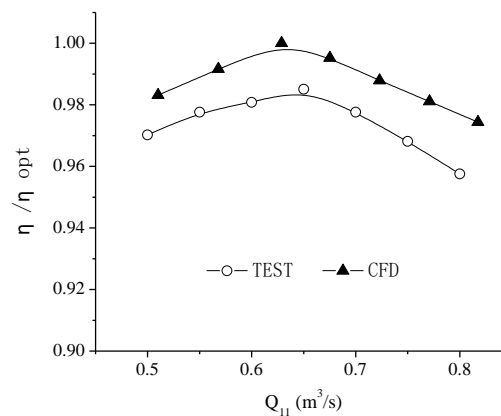
To achieve a favorable flow in draft tube, the runner outlet velocity distribution is highly important. It can not only improve the energy recovery of the draft tube, but also reduce the pressure pulsations induced by the draft tube vortex [11]. Runner outlet flow distribution is one of the most important

characters in runner hydraulic design and critical in improving the turbine efficiency. The simulation results prove that the new runner designed by DFEM could adapt the existed draft tube completely, as shown in figure 10 and 11. The loss of draft tube is almost 0.6% and 1.8% in optimum and rated condition.

#### 4.6 Turbine efficiency

The final results show that the hydraulic loss of spiral casing and stay vanes increase from part load to full load condition, which is opposite of guide vanes, and the loss of runner and draft tube is often the minimum near the optimum point. This tendency of the calculated results is in line with the practical experience [12].

The predicted efficiency by CFD is a little higher than the test data, as shown in figure 12. In addition to the numerical uncertainties, this is also due to the truth that some losses measured in the experiment, e.g. the volumetric loss, disc frictions and mechanical loss etc., are not considered in the CFD calculation. However, the trend of turbine efficiency obtained from CFD agrees fairly well with the model test. This proves that the CFD simulation could predict the efficiency with certain accuracy, which is practically useful in hydraulic development of turbine.



**Figure 12.** Comparison of the Calculation and Tested efficiency

## 5 Model test results

**Table 4.** Main results from model test

Item	1-3# unit	4-6# unit
Model optimum efficiency $\eta_{opt}$ (%)	$\geq 93.5$	$\geq 93.5$
Prototype optimum efficiency $\eta_{opt}$ (%)	$\geq 95$	$\geq 95$
Model rated efficiency $\eta_r$ (%)	$\geq 92$	$\geq 92$
Prototype rated efficiency $\eta_r$ (%)	$\geq 93.7$	$\geq 93.7$
Model weighted average efficiency $\eta_{ave}$ (%)	$\geq 93$	$\geq 93$
Prototype weighted average efficiency $\eta_{ave}$ (%)	$\geq 94.5$	$\geq 94.5$
Pressure pulsation in draft cone $\Delta H/H$ (%)	$\leq 5$	$\leq 5$
Prototype runaway speed $n_R$ (r/min)	$\leq 250$	$\leq 230$
Critical sigma at max-output point $\sigma_1$	$\leq 0.055$	$\leq 0.055$

DFEM completed the preliminary and acceptance tests in 2015 on the company's DF-100 Universal Test Rig, whose measuring error is not more than 0.25%. The results proved that the model turbine's hydraulic performance fulfilled all the guarantees.

The main performance values are shown in Table 4. The optimum efficiency of the model turbine is about 93.7%. The incipience of part-load vortex occurred at around 50% percent of rated output. Leading edge cavitation on blade suction and pressure sides is far outside the operating range. The incipient and critical sigma values of rated point can absolutely ensure the cavitation free operation. Runaway speed is also less than the guaranteed value. The flow in the draft tube is relatively stable with the maximum relative amplitude of pressure pulsation less than 5% in the whole operating range.

## 6 Conclusions

The Guri hydraulic turbine development went through several stages of analysis, CFD optimization, model manufacture and model testing. Based on a strong technical know-how of 150m head turbines, DFEM successfully developed the excellent hydraulic models. CFD simulation and model test results prove that the runner developed by DFEM independently has not only high efficiency but also super stability, which would provide a strong guarantee for operation of hydro-generator unit in the future. By starting the design from an existing hydraulic design, both time and cost for Guri development were significantly reduced. And this experience is useful for the development of hydraulic power station rehabilitated in future.

## References

- [1] Huang Y F, Liu G N and Fan S Y 2012 *Research on prototype hydro-turbine operation* (Beijing: Foreign Language Press Co Ltd) p 3-4
- [2] Wang Y L 2002 The discussion on the rated head of Xiaowan power station *J. of Dongfang Electric* **1** p 26-31
- [3] Shi Q H 2010 Hydraulic design of Three Gorges right bank powerhouse turbine for improvement of hydraulic stability *25th IAHR Symp. (Timisoara)* 1755-1315/12/1/012006
- [4] Wang Z N, Luo X Q, Guo P C and Wang Y N 2013 Hydraulic development of the turbine for Gibe III in Ethiopia *The Int. J. on Hydropower & Dams* **2** p 46-50
- [5] Wang Y L, Li G Y, Shi Q H and Wang Z N 2012 Hydraulic design development of Xiluodu Francis turbine *26th IAHR Symp. (Beijing)* IAHRXXIV-231
- [6] Guo P C, Luo X Q and Qin Y C 2006 Numerical performance prediction for a Francis turbine based on Computational Fluid Dynamics stage simulation *J. of Proc. of the CSEE* **17** p 132-137
- [7] Cherny S, Chirkov D, Bannikov D, Lapin V, Skorospelov V, Eshkunova I and Avdushenko A 2010 3D numerical simulation of transient processes in hydraulic turbines *25th IAHR Symp. (Timisoara)* 1755-1315/ 12/ 1/ 012071
- [8] Li J, Yu J X and Wu Y L 2010 3D unsteady turbulent simulations of transients of the Francis turbine *25th IAHR Symp. (Timisoara)* 1755-1315/ 12/ 1/ 012001
- [9] Wu Y, Liu S, Wu X, Dou H, Zhang L and Tao X 2010 Turbulent flow computation through a model Francis turbine and its performance prediction *25th IAHR Symp. (Timisoara)* 1755-1315/ 12/ 1/ 012004
- [10] Shi Q H and Stuart Coulson 2005 Hydraulic design of Three Gorges left bank powerhouse turbine *Dongfang electric review* **4** p 169-186
- [11] Wang F J, Liao C L and Tang X L 2012 Numerical Simulation of Pressure Fluctuations in a Large Francis Turbine Runner *Chinese J. of Mechanical Engineering* **6** p 1198-1204
- [12] Yan Z G, Zhou L J and Wang Z W 2012 Turbine efficiency test on a large hydraulic turbine unit, *Science China(Technological Sciences)* **8** p 2199-2205