**Proceedings of FEDSM2008**
**2008 ASME Fluids Engineering Conference**
**August 10-14, 2008 Jacksonville, Florida USA**

**FEDSM2008-55245**

# The Alteration of Runner and Partial Vanes on a Fixed Blade Propeller Water Turbine Basing on the Full Passage Simulation

**Lanjin Zhang**
**Department of Thermal Engineering**
**Tsinghua University, China**
**zhanglanjin@mail.tsinghua.edu.cn**

**Zhengwei Wang**
**Department of Thermal Engineering**
**Tsinghua University, China**
**wzw@mail.tsinghua.edu.cn**

**Qingqing He**
**Department of Thermal Engineering**
**Tsinghua University, China**
**Hqq03@mails.tsinghua.edu.cn**

**Guangjie Peng**
**Department of Thermal Engineering**
**Tsinghua University, China**
**pgj03@mails.tsinghua.edu.cn**

## ABSTRACT

Basing on the 3D-steady Navier-Stokes equations with standard k-ε turbulence closure models, non-structure mesh with fitted body coordinate and finite element based finite volume method, the internal flow on the full passage of the 6.5-meters head fixed blade propeller water turbine is analyzed. Numerical results show that the low output is caused by unsuitable full passage. The flow on the stay vanes isn't uniform and the circumferential velocity of the runner rim is too large, which leads to a high loss in the draft tube. So the runner and partial stay vanes in the concrete spiral casing are redesigned. The output of the full passage with new runner and new partial stay vanes under 6.5-meters head is 295KW larger than the old one with 240KW output, and the efficiency is 81%, which is larger than former 70%. The redesign of runner and stay vanes is successful.

**Keywords:** Fixed Blade Propeller Water Turbine, efficiency, output, CFD analysis

## INTRODUCTION

A power plant sited in ZheJiang Province entrusted our institution to find out the reasons why their hydraulic turbine's output is lower than the designed output and required us to increase the output of the turbine to 290 KW and its efficiency to 80%. The hydraulic turbine is a fixed-blade propeller one running under a very low head (5 meters to 7.3 meters). When the prototype turbine was produced, the radial dimension of the guide apparatus was shortened, deviating from the model turbine. As a result, the output can not reach the designed rated output. Since the spiral case, guide vanes and draft tube have built in the power plant, only the stay vanes and runner can be replaced. Besides limits above, mass flow of the unit can't be increased, because large mass flow will lead to a much lower head, which will reduce output. From analyses above, we planed improvement methods with following steps: a) Firstly, collect data and background introduction of the water turbine. b) Secondly, suggest an analysis method. c) Thirdly, realize the optimum plan, and redesign the turbine.

Information offered by the power plant is shown followed. The maximum head is 7.3 m and the designed head is 6.5 m. When the guide vane opening is 52 degree with the 6.5 m head, the maximum output per unit is 240

KW, 60 KW lower than the designed output. The diameter of the runner with four blades is 1 meter, and its runner rotation rate is 427.5 r/min. The spiral case is made of concrete, and the stay vanes number is 16. The symmetry guide vanes number is 16, and its height is 400 mm.

Considering the high field test cost, CFD method can well predict the hydraulic character of the water machine [1-2], since the prototype runner has an only one meter diameter. CFD technology is adopted to analyze reasons why the turbine output is lower and to optimize it. The commercial software CFX is used to simulate the inner flow along the whole flow passage of the propeller water turbine including the spiral casing, stay vanes, guide vanes, runner and draft tube, and the Navier-Stokes equations and the standard $\kappa$-$\varepsilon$ turbulent closed model are applied. Through simulating inner flow of the water turbine, its character and flow field, such as velocity, pressure and torque can be concluded. Basing on the inner flow analysis of the old water turbine, a new runner and some new stay vanes is redesigned to perfectly fit the old flow passage yet obtaining the high output required by the power plant.

## NOMENCLATURE

$\eta$ efficiency
H head
P output

## OPTIMUM METHOD AND NUMERICAL CONDITION

This study plans to choose the 6.5 meter head and 52 degree guide vane opening as the analysis and optimum preconditions, because old output under this condition, offered by the power plant, will correct the simulation result. The simulation geometry model of the water turbine includes the spiral casing, stay ring, stay vanes, guide vanes, runner and draft tube, shown in Fig. 1. The mesh element of the spiral casing with stay vanes is 174,000, the guide vanes 1,400,000, the runner 175,000, and the draft tube 50,000.

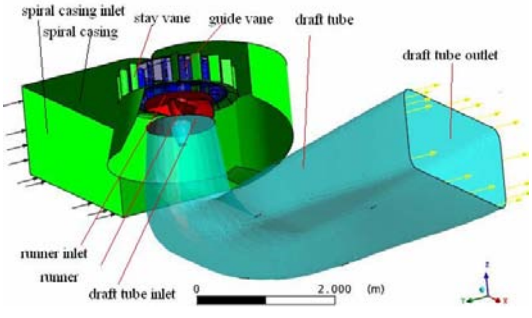


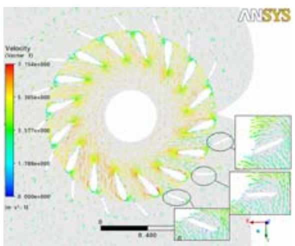

Fig.1 The compute geometry model

The inlet boundary condition of the spiral case is with a total pressure of 67,000 Pa, and the outlet boundary condition of the draft tube is with a nil static pressure. The runner rotates at a 427.5 r/min speed rate, and the condition of the SRI (stator and rotor interface) is stage frozen.

## SIMULATION RESULT OF OLD WATER TURBINE

The simulated output is 240 KW and the efficiency is 70% under the condition of the 6.5 meter head and 52 degree guide vanes opening, shown as the curve named 52_old in Fig.11 and Fig. 10. The simulated loss of spiral casing with stay vanes is 5.6 percent of the head, the loss of guide vanes is 8.3 percent, the loss of runner is 8.3%, and the loss of draft tube is 7.8%. The computed head is 6.54 meter and mass flow is 5.8 $m^3/s$. The output computed matches the power plant's practical output, which shows that the simulation result of the whole passage of the old water turbine is correct. So further flow analysis of the turbine should be done to find out why the output is lower.

The velocity on the spiral casing and stay ring shows that the flow angle of the five stay vanes sited on the spiral casing nose is large enough to cause stall, which strongly affects following flow in the guide vanes, runner and draft tube (seeing in Fig.2). This may be a reason why the hydraulic loss of the stay vanes is that large. Fig.3 shows that the velocity circumferential of the runner inlet near the spiral casing nose is very lower than the others, which is caused by that five stay vanes. Fig.4 shows that those five stay vanes affect the circumferential velocity of draft tube inlet which is same with the guide vane outlet's velocity. So the stay vanes near the spiral casing nose must be redesigned to obtain regular velocity circumferential, which is the first optimal aim.



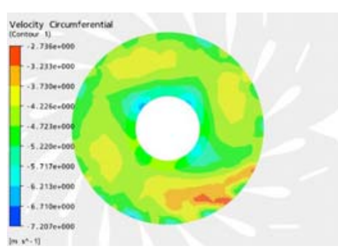




Fig.2 Velocity on old distributor Fig.3 Velocity at old runner inlet

Fig. 4 shows that the circumferential velocity of draft tube inlet isn't affected by that five stay vanes but also by the four runner blades. The average value of velocity circumferential at the draft tube inlet is 1 m/s to 2 m/s, which leads to the high hydraulic loss of the draft tube. In order to verify that, the full axis flow in the draft tube as the inlet condition is simulated separately, and the result shows the hydraulic loss is much lower. So decreasing the velocity circumferential at the draft tube inlet in new water turbine is another optimal aim.

Through analyzing the whole passage flow, it could be concluded that the stay vanes near the spiral case nose lead to the high loss in the flow passage and the large velocity circumferential in the draft tube inlet also increases its loss. So the five stay vanes near the spiral casing nose is redesigned to qualify the spiral casing flow. In order to obtain the axis flow at the draft tube inlet, the runner is redesigned too.

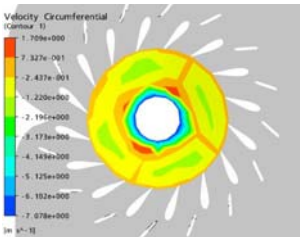



Fig.4 Velocity at old draft tube inlet

## SIMULATION RESULT OF OPTIMUM UNIT

Basing on the flow analysis of the old water turbine and under the condition of 52 degree guide vane opening and 6.5 meter head, the stay vanes near the spiral casing nose (seeing Fig.5), and the runner blade are redesigned.

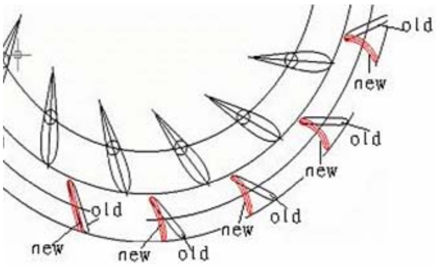



Fig.5 Old and new stay vanes

The out angle and position of those five stay vanes are retained unchanged, but the inlet angle and position are changed. The inlet angle is expressed as below:

$$ctg\delta' = ctg\delta_M + \frac{ctg\delta_e - ctg\delta_M}{360 - \varphi_0}\varphi'$$

Where, $\varphi_0$ is the spiral casing wrap angle, $\varphi'$ is the central angle of the inlet position on the stay vane. $\delta_e$ is the inlet angle of stay vane corresponding to $\varphi_0$, $\delta_M$ is the inlet angle of the stay vane near the spiral casing nose, $\delta'$ is the inlet angle of the stay vane corresponding to $\varphi'$. Once the inlet and outlet are decided, the main curve of the stay vane could be drawn from inlet to outlet by Bezier curve controlled by five points [3]. Then the thickness could be calculated.

The new runner blade number varies from four to five, and the blade is optimized like the stay vanes. But the inlet flow angle of the blade is decided by the axis velocity and circumferential velocity of runner inlet. And the outlet flow angle is decided by axis velocity and rotate speed to obtain nil circumferential velocity. Single blade section is shown as Fig.6.

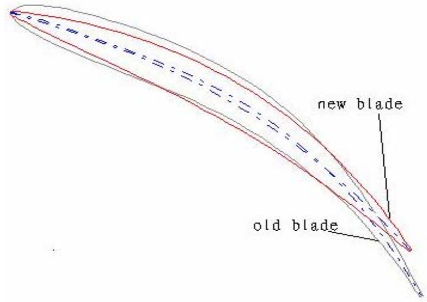



Fig.6 New blade profile and old blade profile

From the curve named 52_new in Fig.12 and Fig.13, it could be seen that the simulated output of the new turbine is 295 KW and the efficiency is 81%, better than the old one. The spiral casing and stay vane hydraulic loss is 2.8%, 2.8% lower than the old water turbine, the guide vane 7.2%, 0.9% lower, the runner 7.4%, 0.7% lower, and the draft tube 1.6%, 6.2% lower. The heavy stall at the spiral casing of the new water turbine disappears (see Fig.7). And the velocity circumferential on runner inlet and the draft tube inlet are also better than the old ones (see Fig.8 to Fig.9). The better draft tube inlet condition leads to a low loss.

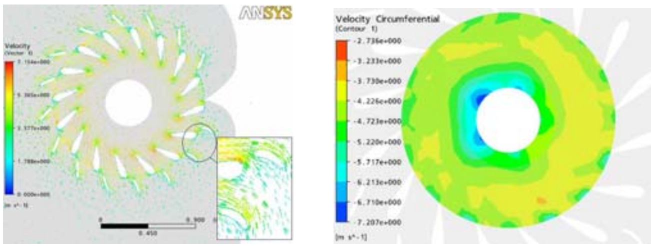



Fig.7 Velocity on new distributor Fig.8 Velocity at new runner inlet

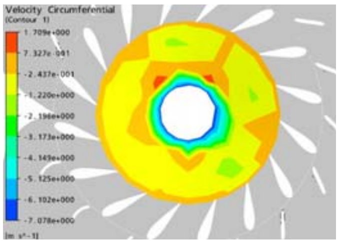


Fig.9 Velocity at new draft tube inlet

## COMPARISON BETWEEN THE OLD WATER TURBINE AND THE NEW ONE

Since only under the condition of 52 degree guide vane opening and 6.5 meter head has the optimum case been proved, more operation conditions of the whole passage should be simulated. Nine operation conditions, listed in the table 1, are computed to compare the characters of the new water turbine and the old one. From the simulating results and analysis, the character curves are obtained, and shown in Fig.10 and Fig.11. Under the condition of the guide vanes opening 52 degree and 7 meter head, the output of the new water turbine is 329 KW and the efficiency is 82%, 12% higher than the old one. And under the condition of the same guide vanes opening and 6.5 meter head, the output of the new turbine is 295 KW and efficiency is 81%. From the point of the efficiency and output, the new water turbine can well satisfy the power plant's requirements well.

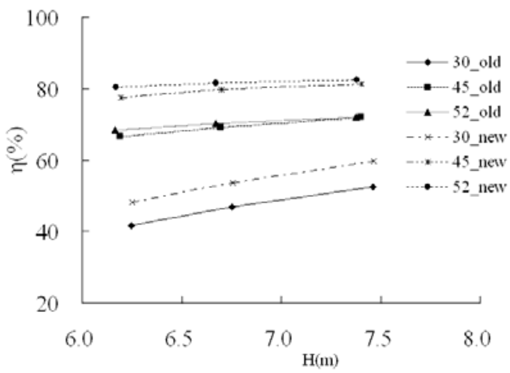



Fig.10 Efficiency-head curve

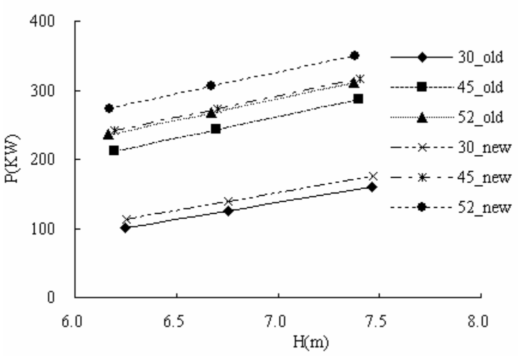



Fig.11 Output-head curve

The velocity on the draft tube section shows that the new flow field will not generate the vortex in the draft tube, yet the old one has two vortexes, shown in Fig.12 and Fig.13. Fig.14 and Fig.15 show the pressure contour of runner blade.

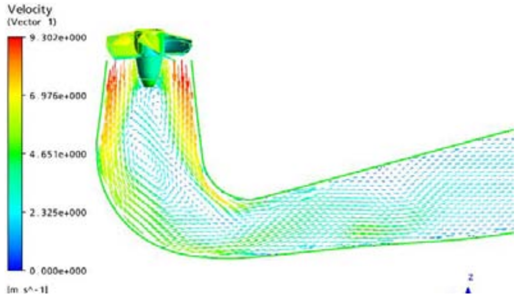


Fig.12 Velocity on draft tube of old turbine

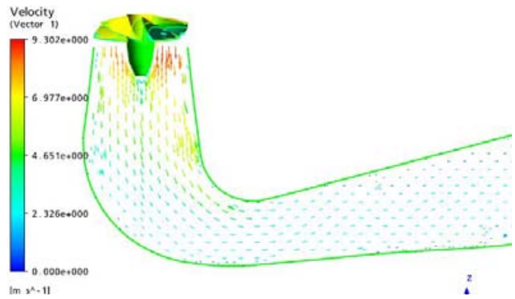


Fig.13 Velocity on draft tube of new turbine

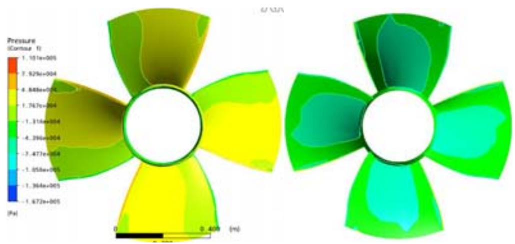


Fig.14 Pressure of old blade

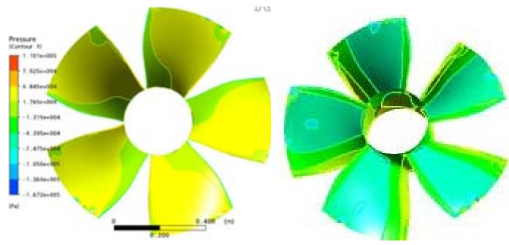


Fig.15 Pressure of new blade

## CONCLUSION

From this study, following results can be concluded.

a) When the fixed blade propeller water turbine runs under a low head, the distributor plays an important role. The outflow of the guide vane should be regular in order to keep the runner can well converting the hydraulic energy into the turbine output efficiently.

b) The inlet flow angle of the new stay vanes near

Table 1 Operation condition points

| Condition points number | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 |
|---|---|---|---|---|---|---|---|---|---|
| Guide vane opening（°） | 30 | 30 | 30 | 45 | 45 | 45 | 52 | 52 | 52 |
| Total pressure inlet($\times 10^4$Pa) | 6.2 | 6.7 | 7.4 | 6.2 | 6.7 | 7.4 | 6.2 | 6.7 | 7.4 |

the spiral casing nose must match the outflow of the spiral casing, otherwise the flow will cause large hydraulic loss and affect the inflow of the runner, like as the old stay vanes.

c) The velocity circumferential of the runner should be low so that the draft tube will generate no vortex and has a low hydraulic loss.

d) The simulation of the whole flow passage of the turbine can provide a better upstream flow boundary for the runner design than the simulation of the separated runner flow passage. Basing on the simulation of the whole flow passage, this case perfectly solves the problem that the runner cannot match with the flow passage well.

e) Through redesigning the five stay vanes and the runner, the output of the new water turbine is 329 KW with the 82% efficiency under the condition of the 7 meter head and the 52 degree. And the output is 295 KW with the 81% efficiency under the condition of the 6.5 meters head and 52 degree. The output can satisfy the power plant's requirement.

## REFERENCE


[1] Jingchun, W., Katsumasa, S., and Kazuo, N., 2007, CFD-based design optimization for hydro turbines, Journal of Fluids Engineering, Vol. 129, No. 2, pp. 159–168.

[2] Wen-Guang, Li, 2004, method for analyzing the performance of centrifugal oil pumps, Journal of Fluids Engineering, Vol. 126, No. 3, pp. 482–485.

[3] XIAO Ruofu, WANG Zhengwei, 2006, Centrifugal pump blade optimization strategy. Journal of Tsinghua University (Sci & Tech), Vol. 46, No.5, 700-703.