CFD ANALYSIS AND REDESIGN OF THE FRANCIS TURBINE WITH LARGE HEAD VARIATION

PENGCHENG GUO[1] , XINGQI LUO[2] and SHENGZHU LIU[3]

It is hard to make a little progress in water turbine stability problem at all times. More attention is paid to this problem in the hydropower plants are of high head and large head variation. The vibration problem always puzzles most of large hydraulic power plants thathave been developed to different extent. The hydraulic problem is one of very important factors to induce vibration. Nowdays flaw and vibration problems baffle the steady operating of large water turbine units to different extent in the whole world. Although these flaw and vibration problems can be caused by many factors, the interaction between the swirling which located in the outlet end of guide vanes resulted from high head and small guide vane openning, and the runner passage vortex and the stall flow caused from the water collision on the inlet of the runner blades is one of the main reasons.

Especially, The Francis turbine in Longtan Hydropower Plant in China is a typical one that is requested to work in the condition of large rated capacity, great variation of water head and huge change of loading. The ratio of the maximal gross water head to the minimal gross water head is 1.588/1.673, and The ratio of the maximal gross water head to the rated water head is 1.112/1.292. Because of the large size of water turbine components, it is extremely hard to be designed and manufactured. Whats more, owing to great variation of water head, huge change of load and large size runner, etc. In Longtan Hydropower Plant, the selection of runner must be carefulness. Not only high efficiency, nice stability and fine cavitation performance of the runner are all demanded, but also much allowance on rigidity and strength was required.

At present, numerical simulation of an entire Francis turbine has been performed,but the fully 3D calculation still requires a lot of CPU time and the hardware costs are expensive. They are all difficult to achieve in the general research agency.So separate and partly coupled numerical analysis were carried out for 22 different points of operation, to modify the model turbine consists of spiral casing with 20 stay vanes, 20 guide vanes, 13-blades runner and elbow draft tube with vertical pier in the paper. Numerical analysis of each turbine component was done in four steps.

At first numerical analysis of spiral casing was performed. The computational domain was extended to the runner outlet, but tandem cascade and runner blades were not modelled. Fluid flow angle was obtained by use of numerical anlysis. The difference value between maximum fluid flow angle and the minimum was about 7 degree. The inlet blade angle of the stay vanes can be acquired from numerical flow analysis for the spiral casing. In theory, changing the stay vanes inlet blade angle to fit spiral casing outlet flow angle, and lessening the collision loss at the front end of the stay vanes and swirl brought by flow separation, can increase turbine efficiency and improve the machine vibration performance resulted from slipstream flow. But the shape of every stay vane will be different if every stay vane inlet blade angle matches the fluid flow angle. This will bring great trouble to manufacturer. In this paper, according to fluid flow angle shown in Fig.1, the stay vanes were divided into 5 groups, namely these stay vanes whose flow angle incidence was less than 2 degree were substituted by a type of stay vane. 20 stay vanes selected were divided into 5 groups.

Secondly, to modify the shape of tandem cascades and alter the relative position of those based on numerical anlysis results, in order to get the blade profile and the relative position of the guide vanes and stay gates, and acquire agreeable results.

Thirdly, flow through stay vanes, guide vanes and runner were calculated simultaneously. Inlet boundary condition was specified from the spiral casing simulation. At the same time, the performance comparison between Negative rake angle blade and conventional blade was done by means of CFD analysis. It was concluded that although the optimized efficiency of the negative rake angle has a little low, the negative rake angle other performances exceed the conventional blade's. In particular, the negative rake angle blades are provided with better operating stability and higher weighted average efficiency. Finally the negative rake angle blade was adopted in the redesigned runner by comprehensive evaluation.

Finally, According to the control sizes in hydropower plant, the model draft tube with an excellent cross section area variation regulation was quoted, and CFD analysis was performed. subsequently in light of CFD analysis results the draft tube were repeatedly modified until the slender draft tube suitable for Longtan Hydropower Plant were obtained.

And then a comparative test was carried out. With respect to the modified turbine, the occurrence of vortex rope under part-load condition and the appearance of stall in blade pressure and suction surfaces were delayed, and the pressure fluctuation amplitude was very low at the spiral casing inlet, guide vanes outlet, runner inlet and draft conical tube upward and backward position etc. At the same time, the redesigned runner possesses high efficiency, the flow separation dosen't occur in the whole flow region and the vortex rope was quite slim and trivial. The redesigned turbine has characteristic of fine stability. In addition, a detailed study of the redesigned turbine based on CFD analysis was performed, and some following conclusions are obtained:

(1) It is valid to use different arrangements of stay vanes to improve the stability and outflow.

(2) It is necessary to optimize tandem cascades to improve the turbine performance on the base of optimizing the spiral casing and stay vanes.

(3) the turbine stability can be advanced by using the negative rake angle blade at low loaded zone.