Research of the cavitation performance of the condensate pump

H F Li, Z B Pan, M H He, K Ji, W C Zhou and S M Min

KSB Shanghai Pump Co., Ltd., 1400 Jiangchuan Rd., Shanghai, 200000, China,

E-mail: shlihaifeng@gmail.com

Abstract. Condensate pump is an important part of power plant circulation systems, which is used to pump condensate water. Because the condensate water pressure is very low, the first impeller of the condensate pump must have a good cavitation performance. Numerical simulation was employed to study the first impeller cavitation performance. The first impeller was set in the condensate pump barrel, and the double suction casing was kept, the parts after the double suction casing was simplified as tube. The simplicity can guarantee the inlet and outlet conditions of the impeller. Based on the RANS and SST $k-\omega$ turbulence model, CFD software was used to simulate the condensate pump at different working conditions. The numerical simulation shows that cavitation occurred at the suction side of the blades closing to the leading edge. The cavitation performance of the impeller was predicted based on the numerical calculation. Comparing with the experimental results, the numerical simulation result is smaller than that of the experiment in small flux, and the cavitation performance trend is agreed with that of the experiments.

1. Introduction

Condensate pump is an important part in power plant circulation system, which is used to pump condensate water from condenser to the deaerator. After doing work, the superheated steam is cooled to condensate water. Because the internal pressure in the condenser is low and the condensate water is nearly in saturation temperature, cavitation is early occurred in the first stage of the condensate pump\[1-3\]. When cavitation occurred, the air dissolved out from the condensate water and mixed with the water vapor to form the mixed bubbles. The bubbles will quickly condense and collapse when it flow to high pressure area. During this process, the complex change of the mass, momentum and energy occur between the bubbles and liquid. Noise, vibration and other physical and chemical phenomena also occur. The cavitation may damage the flow passages $[4-6]$. When the condensate pump cavitation occurs, the pump efficiency will decrease, and what’s more may lead the whole thermal power generator units working unnormal. It is important to study the cavitation to improve the condensate pump anti-cavitation ability. Experiments and numerical analysis are the may methods to study the cavitation. But for large unit, the cavitation experiments are much cost and may not do cavitation experiments because the experimental method limits.

With the development of the computational fluid dynamics(CFD), CFD technology has been used to study the condensate pump phenomena. Feng carried out numerical simulation of the cavitation occurred in the first stage impeller of the condensate pump. He get the cavitation occurring position and analyzed the volume fraction of the liquid and gas within the impeller. Because the physical
model is so simple that it can not simulate the actual working conditions of the first stage impeller, further study is need to validate his simulation.

Because the pressure of the inlet flow is very low, the condensate pump usually was designed as cylinder type. The first stage was set at the bottom of the cylinder to avoid cavitation. The typical type of the condensate pump is as shown in figure 1. The flow at the bottom of the cylinder is very turbulent because of the complex structure of the first stage and its accessories. The first stage of is double suction impeller, the inlet conditions of the up and down are different, so the first stage impeller cavitation is very complex. If feng’s model was used, it can not to simulate the actual inlet and outlet conditions, so it can not to correctly predict the condensate pump cavitation performance.

![Figure 1. The structure of a condensate pump.](image)

In order to correctly analyze the condensate pump cavitation performance, it is important to consider the actual inlet and outlet conditions of the first stage impeller. In this passage, a type of condensate pump NLT500-570 with four stage impellers was studied. The pump was simplified to fit CFD calculations, first of all, the performance of the first stage double suction impeller was predicted, then using two phase model to simulate the cavitation occurred in the double suction impeller, at last, the condensate pump cavitation performance was predicted.

2. Performance prediction
NLT500-570 condensate pump is an important product of Shanghai KSB pump Co.,Ltd.. It has 5 stages; the first stage is double-suction impeller. The double suction impeller has 5 blades. The parameters of the condensate pump are: the designed rotate speed is 1480rpm, the optimum flux is 2250m$^3$/h, the designed head is 87m and efficiency is 83%. The condensate pump(simplified) and the first stage impeller and blades is shown in figure 2.

![Figure 2. Physical model (simplified).](image)

In order to simulate the first stage real working conditions, the calculation area includes the inlet barrel parts, the rotating impeller parts and the double-suction volute outlet parts. Tetrahedral mesh scheme was adopted to mesh the three parts respectively. During the meshing process, the grids were refined in the area where the geometry changes sharply. Figure 3 shows the meshed grid of the calculation area.
In performance numerical prediction, it must to study the mesh sensitivity. Table 1 is the mesh densities used in numerical prediction. The results of mesh sensitivity studies are showed in figure 4. From the figure, it can be found that the good compromise between the results accuracy and the resource requirements is the mesh density $13.8 \times 10^6$. As seen, increasing the mesh density beyond $13.8 \times 10^6$ does not result in a noticeable head increase.

![Figure 3. Numerical analysis grids.](image)

**Table 1.** Mesh densities used in numerical prediction.

| No. | 1    | 2    | 3      | 4       | 5       |
|-----|------|------|--------|---------|---------|
| Grid density | 5E6  | 8E6  | 10.7E6 | 13.8E6  | 15E6    |

![Figure 4. Grid sensitivity test.](image)

Using the tested numerical scheme, the flow within the condensate pump was simulated and the performance of the first stage impeller was predicted. Figure 5 shows the impeller efficiency variation as a function of flux and Figure 6 shows the impeller head variation as a function of flux. In the figure, “Exp” means experimental results and “Num” means numerical simulation results. From the figure, it can be found that the numerical simulation results agree well with the experimental results, this implies that the simplification of the condensate pump and numerical simulation are correct in predicting the first stage impeller.
3. Cavitation prediction

Based on the performance simulation scheme, two-phase flow model was adopted and different inlet pressure was set to simulate the cavitation experiments. But when the two-phase flow model was used and the inlet pressure was set to 1 atmosphere pressure, the numerical simulation can not be convergence. The results analysis showed that cavitation occurred in the second-stage and the blending duct area. Figure 7 shows the water vapor volume fraction in the condensate pump. From the figure, it can be found that when the inlet pressure is 1 atmosphere pressure, the cavitation occurred in the outlet part. Because the second stage impeller was not considered, the simplified physical model may not meet the cavitation simulation requirements.

In order to carry out cavitation simulation, the physical model was simplified for the second time. Keeping the inlet part and the impeller parts do not change, the double-volute outlet part was re-
The double-volute was reserved to keep the outlet conditions of the double impeller do not change. The secondary stages volutes are simplified as straight pipe. Figure 8 shows the simplified outlet parts. After this simplification, the condensate pump performance was predicted using the same numerical scheme. The predicted performance is shown in figure 9 and figure 10. In the figures, “Num” means the first simplification results, “Num2” means the second simplification and “Exp” is the experimental results. From the figures, it can be found that the second simplification do not cause significant difference when used in numerical simulation. The second simplification was used to simulate the condensate cavitation performance.

Figure 8. The second simplification of outlet part.

![Figure 8](image)

![Figure 9](image)

Figure 9. Impeller head variation as a function of flux.

![Figure 10](image)

Figure 10. Impeller efficiency variation as a function of flux.
Under the optimum working conditions, the condensate pump cavitation performance was simulated. The simulation shows that when the inlet pressure is below some extent, the cavitation occurred in the area of the suction side close to the leading edge. Figure 11 shows the water vapor volume fraction and the pressure of the blades. It can be found that at the suction side close to the leading edge, the pressure is low, and then in this area, water vapor area appears. The water vapor volume fraction distribution is similar to the pressure distribution. This shows that the cavitation is closely related to the pressure.

![Figure 11. Water vapor volume fraction and pressure distribution of the blades.](image1)

Figure 12 shows the water vapor fraction distribution in the simplified model. It can be found that the water vapor volume fraction in the condensate pump is nearly zero just except the impeller area. This means that the simplified model can be used to simulate the cavitation simulation.

Based on the simplified model and numerical scheme, the inlet pressure was decreased to model the cavitation process under the optimum working conditions. The head variation as a function of cavitation is shown in figure 13. From the figure it can be found that the impeller head changes with the cavitation decreasing. When the cavitation is below a certain value, the impeller head decrease to be below 97% impeller head when working under normal atmosphere pressure. The certain value is deemed as the impeller cavitation. From the figure, it can be conclude that the cavitation of the impeller under the optimum working conditions is about 6.81m.

Figure 14 shows the evolution of the water vapor in the impeller with the inlet pressure changing. With the inlet pressure decreasing, the water vapor volume increase. The pump performance drop sharply when the water vapor increases to some extent to block the flow passage of the impeller when the inlet pressure decrease.
Using the same procedure, the cavitation of the impeller was obtained when the condensate pump working under different flux. Figure 15 gives the impeller cavitation variation as a function of flux. In the figure, “Exp” means experimental results and “Num” means numerical simulation results. From the figure, it can be found that the cavitation obtained by the numerical prediction agrees well with that of the experimental results. It can also be found that the impeller cavitation is small at low flux, this means that the pump anti-cavitation capacity is good when it working in small flux. With the flux increasing, the impeller cavitation increasing significantly, this implies that the pump anti-cavitation capacity needs to be improved. Because the condensate pump working in a wide flux range, the barrel height must be high enough t needs to ensure the water in the barrel must be higher than the impeller cavitation.

![Figure 13. Head variation as a function of cavitation.](image)

![Figure 14. Water vapor in the impeller.](image)

![Figure 15. Impeller cavitation as a function of flux.](image)
4. Conclusions
Based on the CFD technology, the flow within the condensate pump was simulated and analyzed. The performance of the first stage impeller was predicted and compared with that of the experimental results. Then the condensate pump was re-simplified to carry out cavitation numerical calculation. The water vapor evolution and the cavitation performance of the impeller were predicted.

The pump performance and cavitation performance predicted by the numerical simulation agree well with that of the experiments. It is necessary to carry out some simplification of the condensate pump to meet the numerical simulation.

References
[1] Fen J H, Yang C X, Han B H and Wei P R 2008 *Journal of Lanzhou university of technology* 34(5) 55-59
[2] Li S H 2006 *Thermal power generation* 7 70-71
[3] Chen X and Tan H 2008 *Electrical Machinery Technology* 6 42-44
[4] Wu C S, He S L, Zhu D X, et al 2006 *Journal of Hydrodynamics, Ser B* 18 (3) 217-222
[5] Dupont P 2003 *International Journal of Rotating Machinery* 9 163-170
[6] Rhee S H, Kawamura T and Li H Y 2005 *Journal of Fluids Engineering* 127 986-994