Investigation on cavitation for hump characteristics of a pump turbine in pump mode

D Y Li1, H J Wang1,3, G M Xiang1, R Z Gong1 and X Z Wei1,2
1 Harbin Institute of Technology, School of Energy Science and Engineering, Harbin 150001
2 Harbin Institute of Large Electric Machinery, Harbin, 150040
E-mail: wanghongjie@hit.edu.cn

Abstract. In order to get the accurate hump characteristic curve of a pump turbine in pump mode, three dimensional steady simulations were carried out using SST k-ω turbulence model with cavitation model and without cavitation model under different operation condition points with 19 mm guide vanes opening. A refinement grids were generated to adapt the turbulence model. The results obtained with cavitation model show a better agreement with experiments. The detailed analysis was undertaken to find out the relationship between the cavitation and hump region. It is concluded that the hump characteristic is related with cavitation.

1. Introduction
Cavitation in rotating machinery is a well know phenomenon that occurs when local pressure of the working fluid approaches vapour pressure. This phenomenon is usually observed in areas of the high-speed fluid machinery such as propellers and pumps, where the flow accelerates and the pressure decreases [1]. In such a situation, there could appear strong unsteady forces, noise and erosion. These become a cause of serious accidents involving fluid machinery. Many experiments have been done to investigate the physics of cavitation. And lots of numerical simulations have been carried out to study the cavity development inside rotating machinery.

Since the pump turbine would be switched frequently between pump mode and turbine mode, even with a few minutes, the stability shows more important than that in other rotating machinery. Hump characteristics is one of the special characteristics of a pump turbine at pump mode with small flow. Especially, when the pump turbine runs at high level head, the hump region is unavoidable and the performance is reduced [2].

Lots of research has been carried out for investigation of stability of a pump turbine at pump mode. Braun [3] analysed flow phenomena in details related to the unstable energy-discharge characteristics of a pump turbine at pump mode, he pointed out that the sudden change in the global performance parameters is related to a change of the secondary flow pattern in the diffuser channels. Yin [4] predicted performance in the vaned distributor of a pump turbine under low flow rate in pump mode and concluded that the hump characteristics can be ascribed to the special hydraulic loss characteristics of the stay vanes and guide vanes. Less information is available on the relationship between cavitation and hump characteristics. Liu [2, 5] conducted that the hump characteristics might be attributed to the appearance of cavitation at the inlet of suction side of the runner using an improved
cavitation model. Anciger [6] predicted rotating stall and cavitation inception in pump turbines. The analysis of hump characteristics combined with cavitation phenomenon still need be undertaken. In this paper, three dimensional steady simulations are carried out using SST $k-\omega$ turbulence model with cavitation model and without cavitation model under different guide vanes opening in order to analyze the relationship between hump characteristics and cavitation phenomenon.

2. Pump turbine modelling and numerical method
The simulation model includes draft tube (inlet), runner, guide vanes, stay vanes and spiral casing (outlet) as shown in Figure 1. Geometry configuration was established using the UG commercial software. The parameters of the reduced scale pump turbine are listed in Table 1. 19mm guide vane opening operation conditions were chosen for simulations.

![Figure 1. Computational domain](image)

Table 1. Parameters of the reduced scale pump turbine (pump mode)

| Parameters                              | Values |
|-----------------------------------------|--------|
| Nominal diameter of runner at inlet (mm)| 274    |
| Nominal diameter of runner at outlet (mm)| 524    |
| Number of runner blades                 | 9      |
| Number of stay vanes                   | 20     |
| Number of guide vanes                  | 20     |
| The height of guide vane (mm)           | 45.77  |

Structured hexahedral and unstructured (tongue as for its sharp shape) grids were generated for each part employed ANSYS ICEM. In order to capture the flow separation of guide vanes, stay vanes as well as runner using SST $k-\omega$ turbulence model, the mesh layers are more than 10 and $y+$ at the wall layer is less than 2. The grids for different parts are shown in Figure 2. The total nodes are 8 million.

In this paper, the SST $k-\omega$ turbulence model was used to close the Reynolds averaged Navier-Stokes (RANS) solver for solution of the two phase turbulence flows. The Rayleigh Plesset model was used to model cavitation in the pump turbine. The homogeneous multiphase model with water and water vapour two fluids was employed without consideration thermal phase change. In this simulation, Static pressure at draft tube inlet was specified at pump mode, and the intitial value of liquid water and water vapour was 1 and 0, respectively. The vapour pressure was set 2693 according to the exprimental data. Mass flow was imposed at the casing outlet. No slip wall boundary condition for the solid walls was used. High Resolution was chose for both advceition scheme and turbulence numerics. The Root Mean Square (RMS) was imposed as convergence criterion. The single phase steady flow calculation was set for initialization condition to improve the convergence rapidity.
3. Analysis of external characteristics curve

19mm guide vanes opening was chosen for simulation. Figure 3 shows that the simulation results using SST \( \kappa-\omega \) turbulence model with cavitation and without cavitation model. The discharge coefficient \( \varphi \), head coefficient \( \psi \), and torque coefficient \( \lambda \) are defined as follows.

\[
\varphi = \frac{Q}{\pi R_1^2 \omega} \tag{1}
\]
\[
\psi = \frac{2gH}{R_1^2 \omega^2} \tag{2}
\]
\[
\lambda = \frac{M}{\pi R_1^5 \omega^2 \rho} \tag{3}
\]

where \( R_1 \) stands for radius of runner inlet (pump mode), m. \( Q \) is volume flow rate, \( m^3 \cdot s^{-1} \). \( \rho \) stands water density, \( \text{kg} \cdot \text{m}^{-3} \). \( H \) is head, m.

Point D is the best efficiency point at 19mm guide vanes opening. It can be seen from Figure 3-a, as for best efficiency point, the difference between single and multiphase model is very little, but the simulation results show a better agreement with experimental data during large discharge, hump region and small discharge region and have the same peak point for head-discharge factor curve. Since the cavitation is more likely to happen during large discharge region of a pump turbine in pump mode, multiphase model results show better than that of single model.

With respect to torque-discharge curve (see Figure 3-b), the simulation results of single model show bigger than experimental data while those of multiphase shows smaller during large discharge region. As for small discharge region the value for both model shows smaller than experimental data. It can also obtained from Figure 3-c that, as for \( F \) large discharge point, the efficiency drop very sharply. The points during the hump region are very near to best efficiency point.
Based on analyzing external characteristics curve, it can be concluded that multiphase model has some advantages to predict the performance of a pump turbine in pump mode.

![Figure 3](image)

**Figure 3.** Simulation results for head, torque and efficiency

4. Detailed analysis of internal flow

Since multiphase model could show obviously better agreement with experimental data during the large discharge region, F large point was chosen to analyse in detail. Figure 4 shows the comparison flow field within runner for F point. It can be seen that multiphase model could capture the flow separation which leads to cavitation while single model cannot simulate this information. As for head-discharge curve, the values obtained using single model usually are much bigger than experimental data as that single model cannot predict the loss which cavitation flow leads to, especially for pump mode.

Then small discharge point A, hump region wave tough point B and peak point C, large discharge point F were chosen to analyze the cavitation region that appears as flow separation. Figure 5 shows the cavitation region for different operation points. It can be seen that the cavitation region at the large discharge point is largest. The cavitation region of hump region wave tough point B is much larger than that at small discharge region and hump region peak point C, however which is still less than that at larger discharge point. In addition, it can also obtained that the cavitation appears at suction surface of blade inlet in pump mode.

Figure 6 shows the experimental results at the operation B and C during hump region. From the picture, it can be seen that the cavitation region at suction surface of blade inlet at hump region wave tough operation point while the cavitation region is smaller at hump region peak point C. It can be concluded that the hump characteristics has a certain relationship with cavitation. But this analysis shows cavitation influences little. Based this analysis the mechanism of hump characteristics still need to be investigated.
5. Conclusions

In this paper, three dimensional steady simulations were carried out using SST $k-\omega$ turbulence model with cavitation model and without cavitation model under different operation condition points at 19mm guide vanes opening. The results obtained with cavitation model show a better agreement with experiments. It can be concluded that the cavitation model could capture the cavitation phenomenon resulting from separation flow. Then, the detailed analysis is undertaken to find out the relationship between the cavitation and hump region. Analysis shows that the cavitation region appear larger at hump region wave tough point than that at peak point. In addition, the cavitation appears at suction
surface of blade inlet in pump mode for both of simulation and experiments. It is concluded that the hump characteristic has a certain relation with cavitation. Based on this research, a further study could be taken in order to analyze the mechanism.

Acknowledgement
This work was supported by the National Key Technology R&G Program(Project No.2012B AF03B01-X). We thank Harbin Institute of Large Electric Machinery for its test platform.

References
[1] Iga Y, Nohml M, Goto A and Ikohagi T 2004 J. Fluids Eng. 126(3) 419-29.
[2] Liu J T, Liu S H, Wu Y L, Jiao L, Wang L Q and Sun Y K 2012 Computer & Fluids 68 105-11
[3] Braun O, Kueny J L and Avellan F 2005 ASME Fluid Eng. Division Summer Meeting and Exhibition.( Houston, TX, USA, 5-11, November, 2005)
[4] Yin J L, Liu J T, Wang L Q, Jiao L, Wu D Z and Qin D Q 2010 SCIENCE CHINA Techno. Sci. 53(12) 3302-09
[5] Liu J T, Wu Y L and Liu S H 2013 6th Int. Conf. on Pumps and Fans with Compressors and Wind Turbines (Beijing, China, 25-28, November, 2013)
[6] Anciger D, Jung A and Aschenbrenner T 2010 25th IAHR Sym. on Hydraulic Machinery and Systems (Timisoara, Romania, 20-24, September, 2010)