Pump hump characteristic research based on mass transfer equation

D M Liu$^{1,2,5}$, Y Z Zhao$^1$, X B Liu$^2$, Y Ma$^3$ and W F Wang$^4$

$^1$ Research & Development Center of Dongfang Electric Machinery CO., LTD, Deyang, 618000, China.
$^2$ Fluid machinery key laboratory of Xihua University, Chengdu Sichuan 610039, China
$^3$ Guo Dian Da Du River Repair&Installation co., Ltd Leshan, Sichuan 614900 China
$^4$ Sichuan branch of China Hua-Dian Corporation, Sichuan 614900 China

E-mail: liudemin999@126.com

Abstract. The current development of modern pumped storage plants aims towards a higher flexibility in operation, an extended operation range of the hydraulic machine (especially in the pumping mode), and a higher reliability. The pumping requirements are the crucial design drivers, since, even if the turbine mode performance is very sound, the success of a project depends also on the pump turbine delivering the required maximum pump head and starting reliably in pump mode. Pump hump (pump instability working points at highest head) which is an instability source to the pump-turbine vibration is a serious damage to the pump operation on high head. So the pump hump and cavitation number based on the numerical simulation and experiment results are shown in this paper. The pump hump is sensitive affected by the cavitation number. With the cavitation number decreasing, the hump on flow characteristic curve (e.g. head-flow rate curve, $H-Q$ curve) is gradually decreasing until vanished. Predicting cavitating flows with multi-phase CFD computations is still a very challenging task. Some results of ongoing work in this field are presented. The hump on $H-Q$ curve with cavitation number is discussed in this paper.

1. Introduction
The hump characteristic is one of the special characteristics of a pump–turbine at pump mode with small flow. Especially, when the pump–turbine runs at high head level, the hump region is unavoidable and the performance is reduced. If a pump–turbine runs in hump region, the flow is small and it may cause large pressure fluctuations. Strong noise can be heard during the starting period and the start time is prolonged [1]. Nowadays, the stability of a pump–turbine is more and more important, and it has become essential to study the parameters cause instable characteristics. In pump operation, a positive slope of the head-flow curve occurs at operating conditions in which energy transferred to the fluid doesn't contribute to the head rise. This energy dissipates and gives rise to unsteady flow patterns such as vortices.

$^5$ Corresponding author.
For investigation of the stability of pump–turbines, lots of researches have been done. Genter et al. [2] makes the CFD calculation for the pump mode. According to Genter’s research, the onset of fully developed pre-rotation at runner inlet shroud coincides with the instability of the head curve. The calculation shows rotating stall in the stator already in the area of stable operation. Braun et al. [3] analyzed the change in the global performance at pump mode and it was related to a change of the secondary flow pattern in the diffuser channels. Ran et al. [4] studied the unsteady flow in a pump–turbine and presented an improved runner to reduce the hump on $H$-$Q$ curve. The researches on the hump characteristic at pump mode were limited. The mechanism of the hump characteristic was still unknown. The cavitation in a turbo-machinery may reduce the performance of the whole unit. Researches on the cavitation flow in turbo-machineries were a hot topic for the improvement of flow passage components [5-7], but the analysis of hump characteristics affected by the cavitation number have not yet been reported.

Most of recent studies on pump-turbine at pump mode were carried out by single phase, and the reason for causing the hump characteristic is unclear. In this paper, the hump characteristic affected by the cavitation number is tested by the experiment on the test stand and a CFD method with cavitation model is used to analyze the hump characteristic of a model pump-turbine by numerical simulation. The relation between hump and cavitation number is shown. The hump will become weak until vanish with cavitation number decreasing however the hump will restore if the cavitation number decreases to a certain number.

2. Pump turbine specifications
The whole configuration including spiral casing, stay vanes, guide vanes, runner and draft tube is considered in the physical model, as shown in figure 1. The computing domain enables to overcome the influence of boundary conditions, periodic interfaces, and pitch ratio of rotor-stator interface, and it especially considers the non-axisymmetric inflow from the spiral casing.

Specifications of the machine are listed in table 1.

![Figure 1. Computation domain of the pump-turbine model.](image)

3. Mathematical model and computational details
Three-dimensional Reynolds-Averaged Navier-Stokes (RANS) equations are solved in conservative form on structured multi-block grids. A finite volume based discretisation scheme is used, which is up to second order accurate for the convective fluxes and truly second order accurate for the diffusive fluxes. Time dependent computations can be performed with a second order accurate time stepping scheme.

| Table 1. Main characteristics of the tested pump turbine |
|----------------------------------|
| runner diameter at inlet (mm)    | 560 |
| runner diameter at outlet (mm)   | 250 |
| runner blade number $Z_1$        | 7   |
| guide vane number $Z_{01}$       | 20  |
| rotational speed (rpm)           | 1200|
| specific speed (m.kW)            | 91  |
3.1. Governing equations
In homogeneous multiphase model, a common flow field is shared by all fluids, as well as other relevant fields such as temperature and turbulence. Thus, the governing equations of homogeneous model for mass, momentum, volume conservation equation can be written as

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial}{\partial x_j} (\rho_m \bar{u}_j) = 0$$  \hspace{1cm} (1)

$$\frac{\partial (\rho_m \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho_m \bar{u}_i \bar{u}_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \bar{r}_o - \rho_m \bar{u}_i \bar{u}_j \right)$$  \hspace{1cm} (2)

$$\frac{\partial (\rho \alpha)}{\partial t} + \frac{\partial (\rho \alpha u_j)}{\partial x_j} = \rho_l$$ \hspace{1cm} (3)

where:

$$\bar{r}_o = \begin{cases} \mu_m \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) & i \neq j \\ 2\mu_m \frac{\partial \bar{u}_i}{\partial x_j} - \frac{2}{3} \mu_m \nabla \bar{V} & i = j \end{cases}$$

$$\bar{\Phi} = 2\mu_m \left( \frac{\partial \bar{u}_i}{\partial x_j} \right)^2 + \mu_m \left( \frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \bar{u}_i}{\partial x_j} \right)^2, i \neq j$$

$$\rho_m = \alpha \rho_v + (1 - \alpha) \rho_l$$

$$\mu_m = \alpha \mu_v + (1 - \alpha) \mu_l$$

3.2. Cavitation mass transfer model
Singhal et al. [8] has assumed as additional negative pressure drop which equals the half of the turbulence fluctuation \(p_{turb}^*\):

$$p_{turb}^* = 0.39 \rho k$$ \hspace{1cm} (4)

This effect is equivalent to raise the vapor pressure \(p^*_v\) by \(p_{turb}^*/2\) to an equivalent vapor pressure \(p^*_v\).

$$p^*_v = p_v + \frac{p_{turb}^*}{2} = p_v + 0.195 \rho k$$ \hspace{1cm} (5)

Mass transfer model for vapor–liquid mixture based on the theory of evaporation/condensation on a plane surface, and the Kinetic theory of mass transfer are used to get the source terms. To impart the above source terms a consistent appearance between the completely mushy and completely vaporous formulations, the source term of equation (6) is applied only if \(p < p^*_v\). The source term of the Sharp Interfacial Dynamics Model equation (7) is employed, or vice versa. Thus, the source terms \(m\) can be expressed as:
If $p < p^*$

$$\dot{m} = C_1 \frac{3(1 - \alpha_r - \alpha_s)}{r} \frac{2S}{2 - S} \left( \frac{M}{2\pi R} \right)^{3/2} (p^* - p)$$  \hspace{1cm} (6)$$

If $p > p^*$

$$\dot{m} = C_2 \frac{3\alpha_r}{r} \frac{2S}{2 - S} \left( \frac{M}{2\pi R} \right)^{3/2} (p - p^*)$$  \hspace{1cm} (7)$$

3.3. RNG $k-\varepsilon$ model

A turbulence model is needed to close the system of equations which results from the Reynolds averaging. A variety of different turbulence models can be applied depending on the application. The turbulence model is still one of the largest error sources in modern CFD. However, methods with less or even no modeling are even nowadays not applicable on technical problems, so that the RNG $k-\varepsilon$ model is the state-of-the-art for the simulation of complex, three dimensional flows.

3.4. Grid generation

Rotating and stationary domains are matched using transient rotor stator interfaces. The global mesh encompassed 9 million nodes with smallest angles of 20 degrees. As displayed in figure 2, special refinement is applied in the runner (2.4 million nodes) and in the guide vane domain (5.2 million nodes). The mesh independent is validated before the scheme comparison. The $y^+$ on guide vane and runner blade is shown in figure 2(b) and (c), which are smaller than 70. The mesh number can satisfy the calculation requirement according to the comparison between simulation result and experiment result in figure 2(a).

3.5. Boundary condition

3.5.1. Inlet condition. The pressure inlet at the spiral casing (turbine regime) is used, whose profile is assumed to be uniform.

3.5.2. Outlet conditions. Pressure outlet is used at the draft tube (turbine regime), pressure is calculated according to the tail water level.

3.5.3. Other conditions. No-slip condition is assumed on all the solid walls, and standard wall function is used to calculate the turbulence kinetic energy and turbulence dispassion frequency near the wall. The rotation of runner domain is dealt with by means of transient rotator stator method in the unsteady calculation. The time step is 1 degree in the unsteady calculation.

(a) Mesh independent (turbine model) \hspace{1cm} (b) $y^+$ on guide vane \hspace{1cm} (c) $y^+$ on runner blade

Figure 2. Mesh independent and $y^+$ distribution of the pump-turbine model.
4. Calculation result and analysis
The experiment results are shown in the figure 3(a), the \(psi\) (pressure coefficient)-\(phi\) (flow coefficient) curve is obtained at the same guide vane opening and different cavitation number (sigma number). The curve is different at different cavitation number. When the cavitation number equals 0.18, the hump characteristic is evident at the \(phi = 0.212\). When the cavitation number equals 0.15 and 0.12, the hump characteristic becomes weak and died away respectively. However, when the cavitation number equals 0.09, the hump characteristic becomes evident and appears ahead of time than the cavitation number equals 0.18.

The comparison between experiment results and numerical results is shown in the figure 3(b) and (c). According to the results at cavitation number equals 0.18 and 0.09, the difference between experiment results and numerical results is in a reasonable range. The pressure coefficient at the numerical results is smaller than the experiment results. The hump position is nearly same between calculation result and experiment result.

The hump characteristic (named first hump) on \(sigma = 0.18\) is caused by the guide vane inner part vortex shedding. A large number of vortexes block the guide vane flow passage from runner to spiral case. The vortex gradually appears on the guide vane flow passage when the flow coefficient decreases from working point A to C in figure 4. The centrifugal force is pushes aside by the pressure because there is not enough water to fill with the flow passage.

A little number of cavitation bubbles appears on the runner suction face which is possible reason to hump becomes weak and appears ahead of time. When the flow discharge decreases, the flow impacting becomes serious. So the cavitation bubble appears on two blades on working point C.

From the figure 6, the pressure distribution on runner suction face. On working point D, the absolute pressure is larger than the vapor pressure, so the cavitation bubble can’t appear on runner suction face. On working point B, the absolute pressure is smaller than the vapor pressure, so the cavitation takes place on two runner suction faces. This is the possible reason to the hump appearing on \(psi-fai\) curve [9]. So in this paper, the hump mainly is caused by vortex shedding on guide vane and affected by cavitation number.

5. Conclusions
This paper mainly focuses on the cavitation number affection on the pump’s hump on the smallest flow rate. The hump is compared on different cavitation numbers and same guide vane opening between experiment results and simulation results. By choosing cavitation model, mesh independent validate, hump curve are calculated by the numerical method. The following conclusions can be made:

1) Cavitation numbers make great affection on the pump’s hump shape. The hump will become weak with the cavitation number decreases.

2) The hump can be calculated by the CFD method with cavitation model and the results is reasonable compared with the experiment results.

3) The hump on same guide vane opening appears ahead of time with the cavitation number decreasing.

Acknowledgments
This work is supported by the Chinese National Foundation of Natural Science (No. 51279172), the Open Research Funded Project of Key Laboratory of Fluid Machinery in Sichuan Province (Xihua University) (Grant No.Szjj2012-041, szjj2011-039).
Figure 3. Psi-phi curve on different cavitation numbers.

Figure 4. Vortex sheds on guide vane flow passage at different working points.
Figure 5. Cavitation bubbles appear on runner suction face at different working points.

Figure 6. Pressure distribution on runner suction face at different working points.

References

[1] Mei Z Y 2000 Power generation technology of pumped storage power station (Beijing: China Machine Press) p 200-201
[2] Gentner C, Sallabeger M, Widmer C, et al. IOP Conference Series: Earth and Environmental Science 15 3
[3] Braun O, Kueny J L and Avellan F 2005 Proc. ASME fluids engineering division summer meeting and exhibition (Houston TX USA) pp 1-6
[4] Ran H J, Luo X W and Zhang Y 2008 Numerical analysis of flow phenomena related to the unsteady energy-discharge characteristic of a pump–turbine in pump mode Proc. ASME 2008 Fluids Eng. Division Summer Meeting collocated with the Heat Transfer, Energy Sustainability, and 3rd Energy Nanotechnology Conferences (Jacksonville, Florida, USA, 10-14 Aug 2008) pp 1115-23
[5] Fortes-Patella R, Coutier-Delgosha O, Perrin J and Rebound J L 2007 J. Fluid Eng. 129 128-35
[6] Lindau J W, Kunz R F, Boger D A, Stinebring D R and Gibeling H J 2002 J. Fluid Eng. 124 607–16
[7] Medvitz R B, Kunz R F, Boger D A, Lindau J W, Yocum A M and Pauley L L 2002 J. Fluid Eng. 124 377-83
[8] Singhal A K, Athavale M M and Li H 2002 J. Fluid Eng. 124 617-24
[9] Liu J T, Liu S H and Wu Y L 2012 Computers & Fluids 105-11
Nomenclature

$C_1$ - evaporation coefficient 0.13
$C_2$ - condensation coefficient, 0.01
$C_p$ - Pressure coefficient
$r = 0.0001, S$ is the surface tension
$M$ - torque
$N$ - rotational speed (r/min)
$P$ - output power (MW)
$Re$ - Reynolds number
$p$ - pressure (Pa)
$p_v$ - vapor pressure (Pa)
$u$ - Velocity
$\alpha_v$ - Water vapor void fraction
$\alpha_n$ - non-dissolved gas void fraction, which takes value of 0.0005.
$\sigma = \text{cavitation number (} - C_p)$

$\rho_n$ - mixture density
$\rho_v$ - water vapor density
$\rho_i$ - water density
$\mu_n$ - mixture viscosity
$\mu_v$ - water vapor viscosity
$\mu_i$ - water viscosity
$k$ : Turbulent kinetic. $i, j = 1, 2, 3.$

$\eta = S \frac{k}{\varepsilon}, \eta_n = 4.38, C_{\mu} = 0.0845, \beta = 0.012,$
$C_{is} = 1.42, C_{is_1} = 1.68, \alpha_k = 1.0, \alpha_\varepsilon = 0.769.$

$R$ - Addition term,

$R = \frac{C_v rh^3(1 - h/h_0)}{1 + bh^3} \frac{e^2}{k}$

$\Phi$ - flow coefficient,

$\Phi = Q/(\pi/4D^2u)$

$\Psi$ - pressure coefficient,

$\Psi = 2gH/\left(u^2\right)$