Influence of gravity on hydraulic performance evaluation of an axial flow turbine

H M Lei\textsuperscript{1}, F N Chen\textsuperscript{1}, S H Ahn\textsuperscript{1}, Y X Xiao\textsuperscript{1}, X J Zhou\textsuperscript{2}, C J Zeng\textsuperscript{1}, H G Fan\textsuperscript{1} and Z W Wang\textsuperscript{1*}

\textsuperscript{1} State Key Laboratory of Hydro science and Engineering & Department of Thermal Engineering, Tsinghua University, Beijing 100084, China
\textsuperscript{2} Tianjin tianfa heavy machinery& hydro power equipment manufacture Co., Ltd,& Tianjin Key Laboratory of Tianjin tianfa heavy machinery& hydro power equipment manufacture Tianjin, Tianjin 300405, China

wzw@mail.tsinghua.edu.cn

Abstract. This paper want to analyse the influence of gravity in the axial turbine. In this paper, the influence of axial turbine on the gravity field of different runner diameter is analyzed at the same head, the same opening degree of the guide vanes and runners and the same unit speed. The turbulent model is SST. And in these calculations I consider the impact of runner gap. According to the simulation, when the Fr exceeds 2.3355, the effect of gravity is negligible. When Fr less than 2.3355, the efficiency, flow rate and power considering the gravity are greater than the case without considering gravity. The lower surface of the runner is prone to cavitation when gravity is considered. With the decrease in the diameter of the runner, the difference in pressure distribution between considering the gravity and without consideration of gravity is reduced.

1. Introduction
With the development of the economy and the progress of science and technology, owners of hydraulic design manufacturers put forward the performance requirements such as efficiency, output, anti-cavitation performance and pressure pulsation are getting higher and higher.\textsuperscript{(1)} As a scientific research unit, the hydraulic performance of the hydraulic turbine needs to be accurately evaluated, and the flow field of the turbine is analyzed reasonably. At present, the research on the internal flow of hydraulic turbine is mainly composed of theoretical analysis, experimental study and numerical simulation. With the rapid development of computer technology, CFD numerical simulation with its fast, flexible, short cycle characteristics are widely used by various researchers.

During the study of the turbine flow field, the numerical simulation of the whole flow path from single flow channel, single flow part to turbine. Arakawa\textsuperscript{(2)}, Jacoben\textsuperscript{(3)} and Stantal\textsuperscript{(4)} and so on with the pseudo-compression method to solve the Renault time-averaged equation to calculate the turbine runner turbulence. In the literature\textsuperscript{(5)}, it is considered that the volute and guide vanes should be considered as a whole model in terms of geometric position and from water flow characteristics in order to obtain realistic results. In the numerical study of the internal flow of the hydraulic machinery runner, the numerical simulation of the full three-dimensional viscous flow field has been applied to the motion...
analysis of the turbine overcurrent components from two-dimensional and quasi-three-dimensional development, and people have come to realize that the turbine components are interconnected and interact with each other, and there is a problem of matching between the overcurrent components. It is difficult to achieve the optimal matching between the entire overcurrent components by designing the individual overcurrent components of the turbine. Presently, the analysis of the flow field in the turbine is focused on the guide vane opening\(^6\), the blade form\(^7\) and the different operating conditions\(^8\). As the axial flow turbine head is low, the gravity field on the turbine internal flow field and turbine performance can’t be ignored under certain conditions. Presently, the research on the gravity field in the turbine group is concentrated in the tubular unit, and the numerical simulation of the single-phase fluid in the tidal unit is carried out in the literature\(^9\). The simulation results show that the gravity is closer to the measured result. Therefore, this paper analyzes the influence of gravity field on the hydraulic performance evaluation of axial flow turbine. In the calculation, the clearance of the runner is taken into account, and Fr is differently selected to analyze the influence degree of the gravity field.

2. Numerical Methods

2.1. Control equation

In the turbine, the water as incompressible fluid, is the incompressible three-dimensional turbulent internal flow problem. A large number of large eddy simulation (LES) and Reynolds stress models and the series models based on the Renault Mean Equations (RANS) have emerged in the literature. In the simulation of the prototype, the use of large eddy simulation at this stage is not realistic, mainly large eddy simulation requires dense mesh density, especially in the turbulence active shear zone, inverse pressure gradient region and near wall region, the mesh density required for multi-scale features is unacceptable to the current computer. And the Renault mean equation is the statistical model, the requirement of the computational resources is much smaller than that of the large eddy simulation. Therefore, under the current conditions, the turbulence model based on the Renault mean equation is adopted in this study. And SST \( k-\omega \) model is adopted in the simulations.

The means N-S equation in the restraining coordinate system is as follows:

\[
\frac{\partial}{\partial t} (u_i) = 0 \tag{1}
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_j u_i) = -\frac{\partial p}{\rho \frac{\partial x_i}{\partial x_j}} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_s) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + f_i \tag{2}
\]

Since this study calculates the flow in the rotating runner, the average N-S equation for the Renault in the rotating coordinate system is as follows:

\[
\frac{\partial}{\partial x_i} (w_i) = 0 \tag{3}
\]

\[
\frac{\partial w_i}{\partial t} + \frac{\partial}{\partial x_j} (w_j w_i) = -\frac{\partial p}{\rho \frac{\partial x_i}{\partial x_j}} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_s) \left( \frac{\partial w_i}{\partial x_j} + \frac{\partial w_j}{\partial x_i} \right) \right] + f'_i \tag{4}
\]

\[
\dot{\mathbf{u}} = \dot{\mathbf{w}} + \dot{\omega} \times \mathbf{r} \tag{5}
\]

\[
f' = -2 \mathbf{\omega} \times \dot{\mathbf{w}} - \mathbf{\omega} \times (\mathbf{\omega} \times \mathbf{r}) + \dot{\mathbf{r}} \tag{6}
\]

In this formula \( p \) represents the pressure, \( \rho \) is the density of the water, \( u_i \), \( w_i \) represents absolute
velocity and relative velocity components, \( f_i \) is volume force component, \( \dot{\omega} \) is the rotary angular velocity for the runner, \( \mu \) is the viscosity coefficient, \( \mu_t \) is turbulence viscosity coefficient, and equations are closed by the SST \( k-\omega \) turbulence model.

In this paper, SIMPLEC is used to solve the velocity field and the pressure field in the RANS solution.

3. Physical model and boundary condition
The turbine in this paper is an axial pulsating turbine of a power station. The parameters of axial pulsating turbine are shown in the table below:

| Appellation                                      | Parameter |
|-------------------------------------------------|-----------|
| H/D                                             | 1.36      |
| Design speed n                                  | 125 r/min |
| Number of runner blades Z_1                     | 4         |
| Number of runner Sv Z_0                         | 13        |
| Number of runner Gv                             | 24        |
| Circumscribed circle/ Inscribed circle of Gv    | 1.12      |
| The height of Gv / Circumscribed circle         | 3.13      |
| The number of Middle pier                       | 1         |
| shroud clearance / hub clearance                | 1.5       |

In this paper, Vol means volute; Sv means stay vanes; Gv means guide vanes.

Three-dimensional modeling of axial-flow turbines in UG is shown in figure 1. Turbogrid and Meshing were used to mesh the runner, volute, guide vane and draft tube. In some areas, grid encryption is also done to improve the computational accuracy. The results of the runner meshing are shown in figure 2. The clearance mesh of the runner is shown in figure 3. Through the grid-independent test of the whole calculation domain, the number of meshes is determined as follows: the number of volute mesh is about 720 thousand, the number of draft tube grid is about 860 thousand, the number of Sv mesh is about 182 million, the number of Gv grid about 2.95 million and the number of runner grid is about 1.05 million.
The calculation is carried out by steady calculation. The calculation consists of two conditions the consideration of gravity and gravity does not consider, the calculation of the boundary conditions set as follows:

**Inlet conditions**: Considering the gravity, according to the water turbine head given the total pressure with the relationship of the height; When the gravity is not considered, the total pressure of the inlet is given directly, and the pressure values at each point on the section are equal.

**Outlet conditions**: When considering gravity, using the relationship between the pressure of the outlet at the minimum water level of the tail reservoir with height as the outlet boundary; When the influence of gravity is not taken into account, the total pressure of the outlet is given directly and the pressure values at each point on the section are equal. Consider the gravity and don’t consider the gravity of the inlet and outlet pressure distribution shown in figure 4 and 5.

**Runner speed**: According to the turbine speed given.

**Wall condition**: The solid wall is no-slip boundary conditions.

**Interface**: Since the calculation model is the whole flow channel from the volute inlet to the outlet of the draft, the flow field calculation of each component is carried out at the same time. Therefore, it is necessary to ensure that the velocity component and the turbulence amount coincide at the interface between the parts, and the integral pressure and flow flux consistent. For the static interface and static-dynamic interface should be used in different interface model.

There is no relative motion between the volute and Sv, the Sv and the Gv, and the interface model of None type is used.

The grid of the runner parts is rotated relative to the grid of Gv and the draft part, and the mesh nodes on both sides of the interface do not coincide with each other. The sliding mesh model is used to simulate the static and dynamic interference flow field. The interface between the guide vane and the runner, the interface between the runner and the draft tube is modeled using the Stage rotor model.

**Figure 3.** The clearance mesh of the runner

**Figure 4.** Consider gravity of the inlet and outlet pressure distribution

**Figure 5.** Without consider the gravity of the inlet and outlet pressure distribution
4. Analysis of numerical simulation results

4.1. Influence of Gravity Field on Hydraulic Performance of Axial Flow Turbine

In this paper, the relative opening degree of the guide vanes is 96%, the relative opening degree of the blades is 83%. In order to meet the similarity criteria, the head is 4.5 m in different models, the unit speed is equal in the calculation. In the consideration of gravity and not considering the gravity of the two cases were calculated to analyze the influence of gravity field on hydraulic performance of axial-flow turbine. The calculation results are shown in table 2 and the efficiency is shown in figure 6:

Table 2. The calculation results

| Fr     | η        | Q(m³/s) | Power(kW) |
|--------|----------|---------|-----------|
|        | g  | no_g   | g       | no_g     | g    | no_g     |
| 0.9535 | 86.45%  | 86.12%  | 124.79  | 124.68   | 4745.1 | 4727.3   |
| 1.1677 | 86.09%  | 85.92%  | 55.50   | 55.45    | 2104.6 | 2095.9   |
| 1.3484 | 85.84%  | 85.72%  | 31.22   | 31.20    | 1178.7 | 1176.4   |
| 1.6514 | 84.49%  | 85.18%  | 13.85   | 13.86    | 514.6  | 519.5    |
| 2.3355 | 84.56%  | 84.57%  | 3.47    | 3.47     | 129.0  | 129.0    |

Figure 6. The efficiency in different conditions

In the calculation results, g indicates that gravity is considered in the calculation, no-g indicates that gravity is not considered. It can be seen from table 2 and figure 6, when the head and the unit speed is same, the efficiency of the unit when considering the gravity is larger. With the runner diameter decreases, the deviation of the value gradually reduced. After considering the gravity of the unit flow is larger, which is due to the average flow rate in the inlet is greater than the gravity does not consider conditions. Due to the consideration of gravity flow rate and efficiency are greater than the gravity does not consider conditions, from the power expression can be seen, the power take into account gravity will be greater than when gravity is not considered. With the increase of the diameter of the runner, the influence of the gravitational field is getting smaller and smaller. When the Fr exceeds 2.3355, the influence of the gravitational field is negligible.
4.2. Influence of Gravity Field on the Loss of Axial Flow Turbine Components

In order to further analyze the effect of the gravitational field on the efficiency, the following figure makes three Fr = 0.9535, 1.3484 and 2.3355 considering the total loss distribution of gravity and without considering the gravity six cases. In figure7 the abscissa is the dimensionless Fr, The ordinate is $\Delta h / \Delta h_{\text{max}}$, $\Delta h_{\text{max}}$ represents the maximum value of the total loss in six cases.

![Figure 7. The total loss in six cases](image)

It can be seen from the analysis that when Fr is not more than 2.3355, the total loss of the unit considering the gravity is less than that without adding gravity. In the same head, the gravity field greater impact when the runner diameter is large, the difference in total loss is also greater. As the diameter of the runner increases, the difference in total loss is getting smaller.

In order to analyze the effect of gravity field on the loss of each component, the relative value of each component loss is analyzed. In figure7 the abscissa is the dimensionless Fr, The ordinate is $\Delta h_{\text{component}} / \Delta h_{\text{component max}}$.

$\Delta h_{\text{component}}$ represents the value of the total loss in the component in six cases $\Delta h_{\text{component max}}$ represents the maximum value of the total loss in the component in six cases $\text{Vol}_g$ represents the relative total loss in Vol considering gravity

$\text{Vol}_{\text{no-g}}$ represents the relative total loss in Vol without considering gravity

The naming of other components follows the same rules.
Analysis of the relative loss of each component, we can find out that: When the gravity is considered in the calculation, the loss inside the volute increases as the diameter of the runner increases, and the loss in the stay vanes, the guide vanes, the runner and the draft decreases as the diameter of the runner increases; When the gravity is not taken into account in the calculation, the loss of the components of the unit decreases as the diameter of the runner increases, resulting in an increase in unit efficiency; Compared the loss of components with the same conditions to consider the gravity and do not consider can find out that guide vanes and draft loss considering the gravity are less than no gravity. With reducing the diameter of the runner, the corresponding component loss consideration of gravity and without consideration of gravity tends to be consistent and the efficiency is equal.
4.3. Influence of Gravity Field on Pressure Distribution of Axial Flow Turbine

In order to analyze the pressure distribution of the components in different working conditions, the pressure of distribution \( Fr = 0.9535, 1.3484 \) and \( 2.3355 \) considering gravity and without considering the gravity six cases were analyzed. The relative pressure distribution of the surface of the unit under different operating conditions is shown in the figure 9. The relative pressure is defined as:

\[
P_{rel} = \frac{(P - P_{out})}{\rho gh_{net}}.
\]

It can be seen by observing the pressure distribution on the surface of the unit: the unit surface pressure distribution is more uneven than not considered the gravity at the same head and the same wheel diameter. The surface pressure distribution varies along with the height when considering the gravity of the unit. The volute inlet pressure distribution is not uniform when considering the gravity, the pressure of the bottom is high, the pressure of the top is low, so the import flow rate uneven lead to volute internal loss larger than without considering gravity. The pressure considering the gravity at the draft inlet is lower than without considering gravity. With the decrease of the diameter of the runner, the unevenness of the pressure distribution at the entrance of the volute is reduced, so the influence of the gravity field is reduced and the pressure distribution of the remaining parts is close to the same trend.

As the unit is directly related to the power is the runner and the draft, so the following analyze the runner and draft of the pressure distribution. The selected section is the meridian of the runner shown in figure 10, and also analyze the unit power varies with the diameter of the runner shown in figure 11. In figure 11 the abscissa is the dimensionless \( Fr \), The ordinate is the unit power \( P_{11} \).
Figure 10. The meridian pressure distribution of runner and draft

Figure 11. The unit power varies with the Fr

It is found that the pressure on the upper surface and the lower surface of the runner considering gravity is less than that without considering the gravity. It indicates that the lower surface of the runner is prone to cavitation when gravity is considered. With the decrease in the diameter of the runner, the
difference in pressure distribution between considering the gravity and without consideration of gravity is reduced. The unit power difference of the unit decreases as the diameter of the runner decreases. When Fr exceeds 2.3355, the influence of gravity field is negligible.

5. Conclusions
In this paper, the influence of axial flow pulverized turbine on the gravity field of different runner diameter is analyzed at the same head, the same opening degree of the guide vanes and runners and the same unit speed. The main conclusions are as follows:
Fr more than 2.3355, the impact of gravity field can be ignored.
Fr less than 2.3355, the efficiency, flow rate and power considering the gravity are greater than the case without considering gravity. With the runner diameter decreases, the deviation of the value gradually reduced. The guide vanes and draft loss considering the gravity are less than no gravity.
The lower surface of the runner is prone to cavitation when gravity is considered. With the decrease in the diameter of the runner, the difference in pressure distribution between considering the gravity and without consideration of gravity is reduced. The unit power difference of the unit decreases as the diameter of the runner decreases.

Acknowledgements
Special thanks are due to the National Natural Science Foundation of China (No. 51479093), the State Key Program of National Science of China (Grant No. 51439002), the National Key Research and Development Program of China (No. 2017YFC0404200), and the Economy, Trade and Information Commission of Shenzhen Municipality (201411201645511650) for supporting the present work.

Reference
[1] Yang Li. Flow Field Analysis of Ultra-low Specific Speed Turbine for Cooling Tower [D]. North China Institute of Water Resources and Hydropower, 2010.DOI:10.7666/d.d142761. (in Chinese)
[2] Arakawa C. et al. Numerical Simulation of Francis Runner Using Pseudo-Compressibility [J]. Proc. Of 15th Symposium of the IAHR, Paper CI, 1990, 12(5):15-69
[3] Jacoben O. et al. Three-Dimensional Turbulent Flow Simulation in Hydraulic Machinery[J]. Proc. Of 15th IAHR Symposium, paper C, 1990, 10(3):28-39
[4] Stantal O., et al. Hydraulic Analysis of Flow Computation Results[J]. Proc. Of 16th IAHR Symposium, 1992, 8(2):545-554
[5] Xiaoyuan Zhou, Lunfu Qu, Yulin Wu. Numerical Simulation of Internal Flow in Turbine Spiral Case and Fixed Guide Vane [J]. Journal of Tsinghua University, 2000, 40(8): 93-97 (in Chinese)
[6] Kang Can, Li LiTing, Lu GuoHui etc. Influence of Vane Opening on the Performance and Flow Characteristics of Tubular Turbine [J]. Journal of Hydraulic Engineering, 2016, 34(5):406-413.DOI:10.3969/j.issn.1674-8530.15.0205. (in Chinese)
[7] Zhao Yaping, Liao LiWei, Li ZhiHua. Flow Field Characteristics of Tubular Turbine with Type C and S Type [J]. Journal of Agricultural Engineering, 2013, (17):47-53.DOI:10.3969/j.issn.1002-6819.2013.17.007. (in Chinese)
[8] Li Rennian, Jiang Lei, Li Qifei et al. Analysis of Flow Field in Draft Tube of Pump Turbine Increasing Load [J]. Journal of Hydraulic Engineering, 2015, (1):49-54.DOI:10.3969/j.issn.1002-5335.2015.14.0131. (in Chinese)
[9] Ahn Soohwang, Ahn Soo-Hwang, Xiao Yexiang, Wang Zhengwei*, Zhou Xuezhi, Luo Yongyao. Numerical prediction on the effect of free surface vortex on intake flow characteristics for tidal power station. Renewable Energy. 2017.101, pp. 617-628.
[11] Luo Yongyao, Wang Zhengwei, Xiao Yexiang, He Chenglian. Optimization of the Runner for Extremely Low Head Bidirectional Tidal Bulb Turbine. *Energies*. 2017.10(6)

[12] Soohwang Ahn, Yexiang Xiao, Zhengwei Wang, Yongyao Luo. Numerical prediction on the effect of free surface vortex on intake flow characteristics for tidal power station. *Renewable Energy*. 2017.101, pp. 617-628.

[13] Luo Yongyao, Wang Zhengwei, Liu Xin, Xiao Yexiang, Chen Changkun, Wang Haopin, Yan Jianhua. Numerical prediction of pressure pulsation for a low head bidirectional tidal bulb turbine. *Energy*. 2015, 89: 730–738.

[14] Zhao Xiaoran, Xiao Yexiang, Wang Zhengwei, Luo Hongying, Ahn Soo-Hwang, Yao Yangyang, Fan Honggang. Numerical analysis of non-axisymmetric flow characteristic for a pump-turbine impeller at pump off-design condition. *Renewable Energy*. 2018.115, pp. 1075-1085.