Fifth International Symposium on Marine Propulsors
smp'17, Espoo, Finland, June 2017

A Head Correction Method In Model Experiment of Water-jet Propulsion Axial-flow Pump With Front Guide Vane

Ji Guorui[1,2], Wang Zonglong[1,2] , Cai Youlin[1,2], Li Ning[1,2]

[1]Marine Design & Research Institute of China (MARIC), Shanghai, China;

[2] Science and Technology on Water Jet Propulsion Laboratory, Shanghai, China

ABSTRACT

The water-jet axial pump with front guide vane is a new type of pump. However this kind of pump has the character of low head, so there will be a large deviation in the head measurement because of the experimental pipeline friction loss, in the hydraulic performance testing according to the concerned standard. The experimental result shows that the loss head is more than 5% of the pump fluid head, so the head loss should be paid enough attention. Some research is done in this paper on the friction loss correction in the low head water-jet propulsion pump model hydraulic performance experiment. Three methods are adopted in analyzing the relationship between head loss and flow rate, which are friction loss semi-empirical formula calculation, differential pressure testing, the flow field CFD calculation of the experimental pipeline. A correction formula which is suitable for performance testing of low head standard water-jet propulsion model in laboratory is achieved, by comparing the difference of these three methods. Then the correction method is applied in the hydraulic performance testing of some type of low head water-jet propulsion pump with front guide vane, correcting the head. And the correction method is verified.

Keywords

Water-jet propulsion; Axial-flow pump; Front guide vane; Head;

1 INTRODUCTION

The water-jet propulsion axial-flow pump with front guide vane is a type of pump which is suitable for underwater vehicles and has the characteristic of low head. In the model test, the test tube friction loss has great influence on the pump head, which leads to a big deviation measurement. So it's necessary to modify the head, in order to reflect the performance of the pump accurately. There will be three methods adopted in this paper which are the recommended formula in the ISO 9906 2000, the model pump test, the CFD numerical calculation, to obtain the relationship between the flow and head loss, and according to the data fitting a quadratic equation respectively. By comparing these three kinds of fitting quadratic equation, the appropriate correction method is selected, then the selected method is used to correct a certain type of pump head, which proves the effectiveness of the correction method. It can be used as a guide for the head correction in the same type of water-jet propulsion axial-flow pump with front guide vane.

2 Correction Judgment

First of all, according to the "Figure C.1 Chart for grade 1 tests showing velocities above which loss corrections are required" of the ISO 9906 2000 "Appendix C Friction loss", to determine whether the experiment need to be corrected.

In this paper, the flow rate range of low head pump is 0.3~0.5 m^3/s, and the range of the uncorrected head is 2~8 m. The experimental pipeline is a circular pipe with a diameter of 300 mm, and the flow rate is about 4~7 m/s. As showed in the following figure, the head measurement area falls in the scope of correction required, so it's necessary to correct the measurement head.

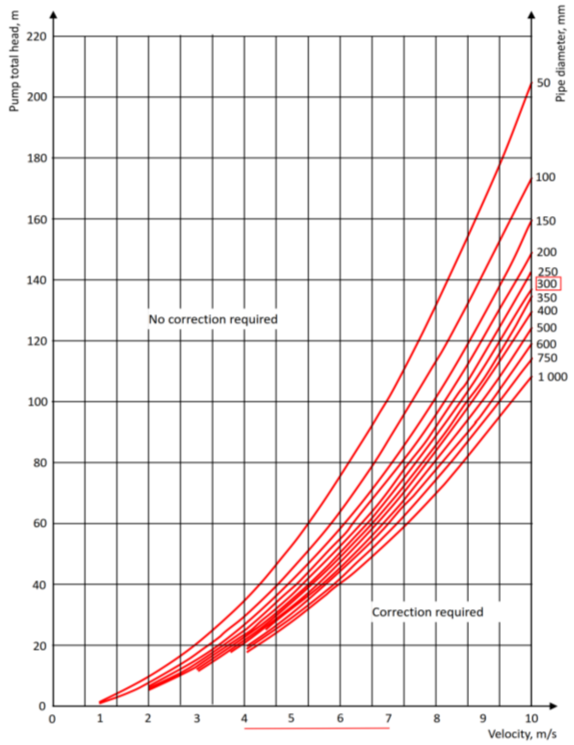

Fig.1 Chart for grade 1 tests showing velocities above which loss corrections are required

3 Test Measurement Correction

The model test of low head pump is carried out by the comprehensive performance test bench. The distance between front and rear pressure measuring points of the experimental pipe and the pump flange is L=600mm. There is a shaft bracket of 100mm length at the end of the outlet flange, as shown in the following Fig.2. The experimental pipe is a circular pipe with diameter of 300 mm.

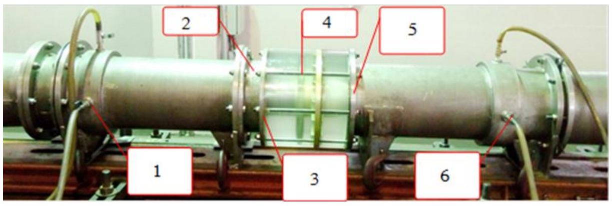

Fig.2 Model pump test section

Where 1=rear pressure measurement point of model pump; 2=shaft bracket; 3=outlet flange of model pump; 4=test pump; 5=inlet flange of model pump; 6=front pressure measurement point of model pump;

The pipe friction loss is measured on the test bench. During this experiment, the model pump is removed, retaining the shaft bracket. Besides, the transparent section is reserved, because of the matching problem between shaft bracket flange and inlet flange. The testing device is shown in the following figure.

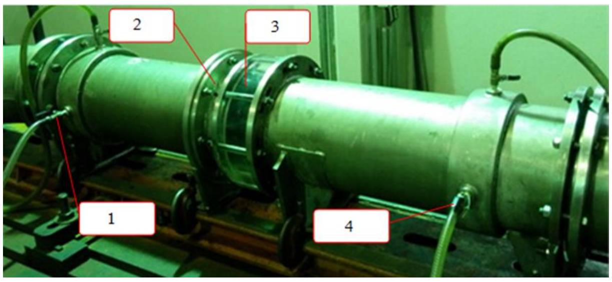

Fig.3 Test section

Where 1=rear pressure measurement point; 2=shaft bracket; 3=transparent section; 4=front pressure measurement point.

In the experiment, the auxiliary pump is turned on, in order to measure the head between the front and rear pressure measurement point under a certain flow rate. And the head loss of the test section can be obtained. The experimental date is showed in the following table.

Tab 1 Head loss gotten by experiment

Measuring Point	$Q(m^3/s)$	H（m）
1	0.37	0.37
2	0.36	0.36
3	0.35	0.34
4	0.34	0.31
5	0.33	0.3
6	0.32	0.28
7	0.31	0.26
8	0.30	0.25
9	0.29	0.23

Restricted by the auxiliary pump power, the flow rate measurement range is 0.29~0.37 m3/s. Beyond the flow rate measurement range, the relationship between the head loss and flow rate is extended through the fitting quadratic curve. The measurement point and the fitting quadratic curve are showed in the following Fig.4.

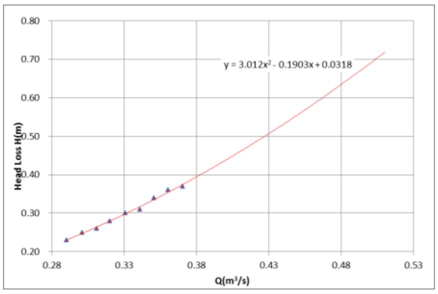

Fig.4 Graph and fitting formula of head loss gotten by experiment

4. CFD Calculation Correction

The 3D model of the model pump experimental section is established before calculating the model pump experimental section friction loss by CFD method. The size of the established 3D model of the model pump is consistent with the test model, which includes the shaft, propeller, etc. The 3D model is showed in the following Fig.5.

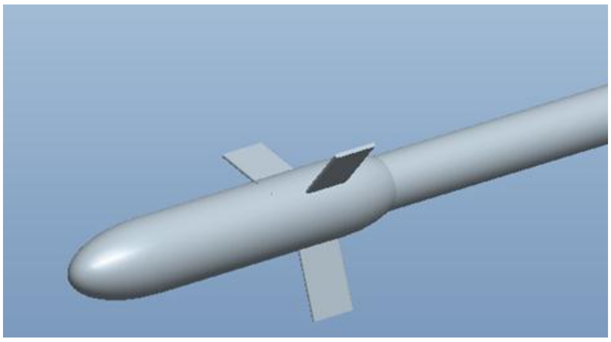
Fig.5 3D model of test section

Before the actual calculation, many models with different grid number are calculated, in order to eliminate the grid sensitivity. Finally, the prism and tetrahedral hybrid mesh is adopted in meshing the 3D pump model, and the y+ is 60. The total number of grids is about 528 million, as showed in the following figure.

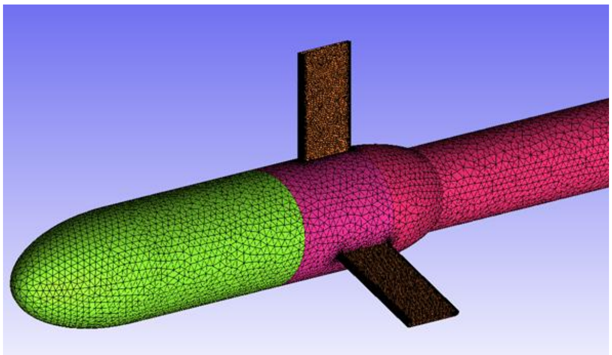
Fig.6 3D grid of model pump

Then the mesh is imported into the software Fluent, the k-ω SST turbulence model and couple method is adopted. The inlet boundary condition is set to be PressureInlet, the outlet condition boundary is set to be mass flow inlet. The CFD numerical calculation is not same as the model test that the range of flow rate is restricted, so the extend of the flow rate is bigger than the model test. The calculation result is showed in the following table .

Tab.2 Head loss calculated by CFD

Calculation Point	Q（m^3/s）	H（m）
1	0.51	0.63
2	0.49	0.59
3	0.47	0.54
4	0.45	0.5
5	0.43	0.46
6	0.41	0.42
7	0.37	0.35
8	0.35	0.31
9	0.33	0.28

The above data is drawn into a graph and fitted with quadratic function, as showed below:

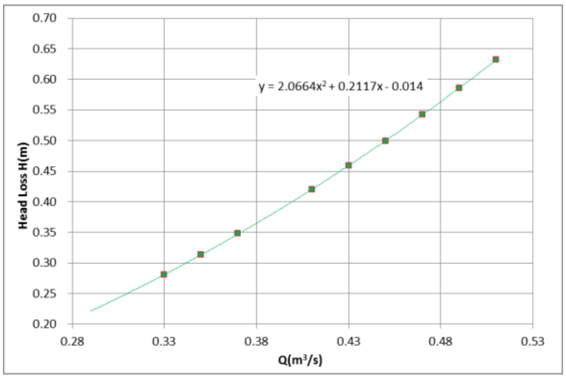

Fig.7 Graph and fitting formula of head loss calculated by CFD

5. Theoretical Formula Calculation

According to the standard ISO 9906 2000, the length between the calculation point and flange is L, and the pipe is a circular pipe with the fixed radius, so the friction loss can be computed by the following formula：

$$H_\lambda = \lambda \frac{L}{D}\frac{U^2}{2g} \tag{1}$$

$$\frac{1}{\sqrt{\lambda}} = -2\log_{10}\left[\frac{2.51}{\mathrm{R}_e\sqrt{\lambda}} + \frac{k}{3.7D}\right] \tag{2}$$

Where: k=is the pipe equivalent uniform roughness; λ =friction loss coefficient; D= is the pipe diameter; k/D=is the relative roughness.

The formula (2) is converted to the get the friction loss coefficient expression of the straight pipe, as showed below:

$$\lambda = \left(\frac{1}{\frac{-2}{\ln 10} - \frac{R_e \cdot k}{2.51\cdot 3.7\cdot D}}\right)^2 = \left(\frac{2.51\cdot 3.7\cdot \ln 10}{2\cdot 2.51\cdot 3.7 + \frac{4\cdot k\cdot \ln 10\cdot Q}{\pi\cdot\gamma\cdot D^2}}\right)^2 \tag{3}$$

The length and diameter of the experimental pipe is L＝1.31m，D＝0.3m. the pipe equivalent uniform roughness is 12.5×10-6m, then the straight pipe head loss of different mass flow rate can be obtained through the formula (3); The shape loss of the shaft bracket can be calculated as the local head loss through the formula (4), as showed below:

$$H_j = \zeta\frac{V^2}{2g} \tag{4}$$

Where: ξ =The local resistance coefficient; According to the valve loss coefficient, ξ is 0.13; V=Velocity of the water in the shaft bracket; The flow area of the shaft bracket S＝0.0602718 m2, so the V can be calculated.

So the head loss of different flow rate can be computed through above formula:

Tab.3 Head loss calculated through formula

Q (m^3/s)	V (m/s)	H_λ	H_j	H
0.51	7.2150	0.1337	0.4749	0.6086
0.49	6.9321	0.1327	0.4384	0.5711
0.47	6.6491	0.1316	0.4033	0.5349
0.45	6.3662	0.1305	0.3697	0.5002
0.43	6.0833	0.1292	0.3376	0.4668
0.41	5.8003	0.1278	0.3069	0.4348
0.37	5.2401	0.1248	0.2505	0.3753
0.35	4.9515	0.1230	0.2237	0.3467
0.33	4.6685	0.1211	0.1988	0.3199
0.31	4.3856	0.1190	0.1755	0.2944
0.29	4.1027	0.1166	0.1536	0.2702

6. Comparison and Verification of Three Methods of Head Loss Correction

6.1 Comparison of Head Loss Correction Methods

In order to observe the relationship between flow rate and friction head loss of three methods, the date attained through three methods are plotted in one figure, as showed below:

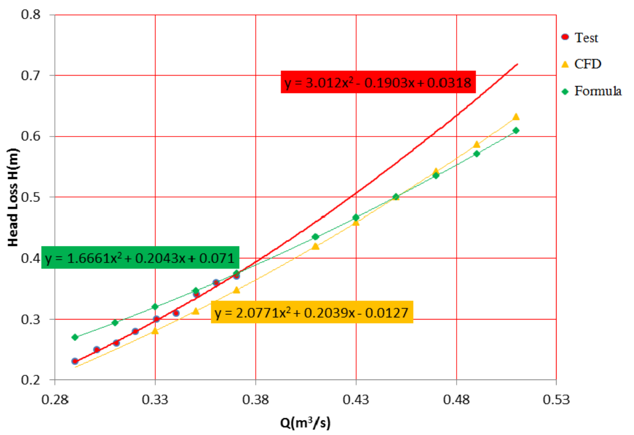

Fig.8 Comparison of head loss gotten by three methods

In the figure above, the red line with dots is gotten by experiment. The yellow line with triangle points is calculated by CFD. The green line with diamond points is attained by semi-empirical formula. From the figure, we can conclude that the relationship between flow rate and head loss obtained by three methods is basically consistent. In the small mass flow range, the friction loss calculated by CFD is close to that of experimental measurement. In the bigger flow rate range, the friction loss obtained by CFD is close to that of theoretical formula. Compared with three curves of the above figure, the head loss curve obtained by the CFD calculation is more conservative, which is beneficial to the engineering design, besides the correction fitting equitation is easy to get compared with the other two methods. The friction loss fitting curve attained by CFD method is showed as below:

$$y = 2.077x2 + 0.203x - 0.012 \qquad (5)$$

6.2 Experimental Verification

The test of a certain type of pump is conducted, the flow rate is Q=0.44 m ³/s at the design efficient point, and the design efficiency is 0.85, measuring the head and efficiency of different flow rate, then correcting the head and efficiency through the formula (5). The correction result is showed as below:

Tab.4 Experimental and correction data

$Q(m^3/s)$	H(m)	η(%)	$H_{-Co}(H)$	$\eta_{-Co}(\%)$
0.4758	3.27	71.47	3.8248	83.76
0.4700	3.46	72.52	4.0022	84.01
0.4609	3.82	74.36	4.3428	84.63
0.4507	4.12	75.72	4.6214	85.08
0.4404	4.43	76.62	4.9102	85.07
0.4307	4.75	76.76	5.2107	84.23
0.4204	5.03	76.55	5.4704	83.34
0.4102	5.23	76.32	5.6508	82.51

The result of the above table shows that the experimental efficiency is lower because of the pipe friction loss. The efficiency correction range is 6% ~ 12%. And the efficiency is 76.62%, when the flow rate is Q=0.44m ³/s, the efficiency is corrected to be 85.08% by the correction formula (5), which verifies the validity of the CFD correction method.

7. Conclusion

According to the above calculation and comparison, we can draw the conclusion as below,

1. The low flow rate water-jet propulsion pump has the characteristic of low head, and the friction loss take a great part of the loss, so the head of this kind of pump should be corrected when the model test is conducted.

2. After experimental verification, it proves that the head correction formula attained through CFD method can be used as correction formula for low head model pump with front guide vane.

REFERENCES

Dai Yuan-xing, Wang Li-xiang(2013).'General characteristic curve forecast for axial flow pump with front guide vane by CFD', SHIP&BOAT(2013(5)1-5)

Lin Jianzhong, Ruan Xiaodong, Chen Bangguo(2013), Fluid Mechanics.Tsinghua University Press.

Li Zhong, Yang Min-guan(2010),'Numerical Simulation and Experimental Validation of the Flow Field in Axial Flow Pump', Journal of Engineering Thermophysics.2010,31(11).

Standards B. Rotodynamic Pumps - Hydraulic Performance Acceptance Tests - Grades 1, 2 And 3 (Iso 9906:2000).

WANG Chun-lin ZHAO Bai-tong PENG Na SI Yan-lei(2008),'Numerical simulation and performance prediction of interior flow field for axial-flow pump with front guide impeller', DRAINAGE AND IRRIGATION MACHINERY(2008,26(6)).