Proceedings of FEDSM2008
2008 ASME Fluids Engineering Conference
August 10-14, 2008 Jacksonville, Florida USA

FEDSM2008-55144

PREDICTION ON METER FACTOR OF THE TURBINE FLOW METER WITH UNSTEADY NUMERICAL SIMULATION

Gang Chen
Department of Thermal Engineering,
Tsinghua University, Beijing 100084, China
gang-chen05@mails.tsinghua.edu.cn

Yulin Wu
Department of Thermal Engineering,
Tsinghua University, Beijing 100084, China
wyl_dhh@mail.tsinghua.edu.cn

Guangjun Cao
Beijing Petroleum, Oil & Lubrication Institute,
Beijing 102300,China
guangjun_cao@163.com

Mingjie Li
Tianjin Xinke Whole Set Instrument Meter Co.,Ltd
XiQing Development Zone, Tianjin, 300385,China
lmj1956@126.com

Suhong Fu
Beijing Petroleum, Oil & Lubrication Institute, Beijing 102300,China
lwyfsh@163.com

ABSTRACT

The turbine flow meter is widely used in the flow rate measuring for its high accuracy and good repeatability. The flow rate will be calculated based on its meter factor, which is the most important factor of the turbine flow meter. The meter factor means pulses or revolution of the impeller per unit volume, and it can only be got from the calibration experiment. At the given flow rate, the driving torque on the impeller is equal to the drag torque, as many paper have pointed out. Based on the torque balancing equations, unsteady numerical simulation is carried out with RNG turbulence model and UDFs (User Defined Functions) in Fluent Code. The meter factor under different flow rate is calculated with the unsteady simulation. The prediction results based on the numerical simulation showed the same trends as the calibration experiment. At the most flow rate, the meter factor keeps constant, but at the lower flow rate, the meter factor higher than the constant. Because of neglecting the bearing friction drag in the process, the meter factor by numerical simulation is larger than experiment

Keywords: turbine flow meter; meter factor; unsteady numerical simulation;

NOMENCLATURE

T_{dr} driving torque on the impeller, [N • m]

T_i hydraulic drag torque on the impeller, [N • m]

T_{dr} blade tip clearance drag torque, [N • m]

T_b bearing drag torque , [N • m]

T_h hub fluid drag torque , [N • m]

T_{disk} hub disc friction drag torque , [N • m]

K volumetric meter factor of turbine meter, [rpm/L]

K_c volumetric meter factor of turbine meter from numerical simulation, [rpm/L]

K_e volumetric meter factor of turbine meter from calibration experiment , [rpm/L]

Err error between numerical simulation and calibration experiment, [%]

f pulse frequency from the turbine meter, [-]

Z number of impeller blades [-]

N rotational speed of the impeller, [rpm]

M hydraulic moment vector on the impeller [N • m]

J rotational inertia of the rotational system [kg • m^2]

ω angular velocity vector [rad/s]

Q volumetric flow rate, [m^3/h]

ρ density of water, [kg/m^3]

υ kinetic viscosity coefficient of water, [$m^2\,s^{-1}$]

f body force vector [N]

r radius vector from the rotational axis to the mass particle [m]

R vector from the origin to the mass particle

k turbulent kinetic energy [$m^2\,s^{-2}$]

ε turbulent dissipation rate [$m^2\,s^{-3}$]

μ turbulent viscosity coefficient, [kg $m^{-1}\,s^{-1}$]

μ_{eff} effective viscosity coefficient, [kg $m^{-1}\,s^{-1}$]

μ_t vortex viscosity coefficient, [kg $m^{-1}\,s^{-1}$]

INTRODUCTION

For its high accuracy and good repeatability the flow meter is widely used in the flow rate measurement. According to Backer's paper (Backer, 1993), the first modern turbine flow meters was developed in the USA in 1938. The technology in measuring single phase fluid become more and more mature, but many experts want it to be used in multi-phase fluid, such as oil in water or water in oil (A.F. Skea and A.W.R. Hall, 1999). And also the viscosity and the inlet flow condition will influence its precision (Schmidts M, Marliani G, Vasanta Ram VI, 1998 and Backer, 1993).

Theory about turbine flow meter has been proposed for many years, and several mathematical models has been used in the prediction of its meter factor, in which the Thompson's airfoil theory (Thompson, R.E. and Grey, J. 1970) was the basement and has been used by many researchers. Qin Hongxin etc. (1990) calculated the meter factor by Finite Element Analysis method, the results has good agreement with experiment. Y. Xu (1992) developed a model for the prediction of turbine flow meter performance, which based on the aerodynamic characteristics around the blades and considering the bearing friction drag. Recently, as the developing of Cmputational Fluid Dynamics (CFD) technology, also, there are many experts used the CFD in simulate the inner flow of turbine flow meters (Th. Huwener and E. von Lavante, etc, 2001 and Zheng Dandan and Zhang Tao, 2005).

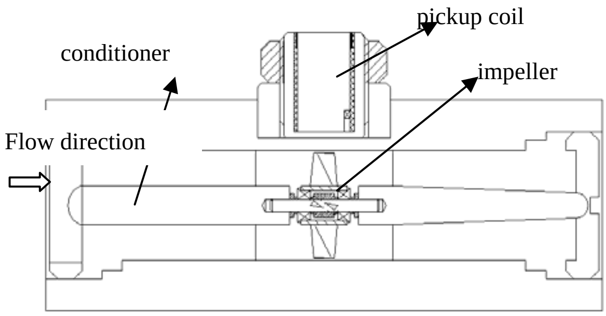

Fig.1 Structure of turbine meter

In this paper, the analysis method of predicting the rotational speed of the impeller is proposed based on the balance equation. With the analysis method, the turbine flow meter is investigated by numerical simulation with CFD subsequently and the meter factor is deduced. Finally, calibration experiment has been carried out on the volume tube systems which can get the meter factor by experiment. The results confirm that the computational method in predicting the meter factor of turbine meter is feasible and effective.

ANALYSIS ON THEORY

The turbine meter consists of a helix type impeller which turns due to the flow in the pipe. In order to hold the impeller in the stream, there are two supports in the upstream flow and downstream flow, which is also called conditioner (Fig.1). The impeller rotates along its axis when fluid flow through the conditioner and impeller. In order to create minimum disturbance when the flow passing around the impeller, its design is different from the pump and turbine. The rotational speed has close relation with the flow rate of the pipeline, which can be obtained by the magnetic pickup.

The meter factor of turbine is described by volumetric meter factor K as follow:

$$K = \frac{f}{Q} \tag{1}$$

Where: f is the pulse frequency, which can be described as the follows:

$$f = z \times n \tag{2}$$

Where: Z is the number of rotor blades and n is the rotational speed of the impeller.

From the equation, the meter factor K is the function of rotational speed, which should be obtained by calibration experiment. Meter factor K is only affected by meter structural parameter; While it is not affected by volumetric flow rate Q, density ρ, ect. So K can be regard as a constant value. But when considering the friction drag and other factors, K shows different value at different flow rate.

When a turbine flow meter is applied in stable conditions (without any unsteady disturbances), all the torques acting on the impeller must balance each other all the time, so the rotor speed remains constant. The balance equation is as follows (Backer 1993):

$$T_{dr} = T_i + T_t + T_b + T_h + T_{disk} \tag{3}$$

Where T_{dr} is the driving torque on impeller,

T_i is the hydraulic drag torque on impeller,

T_t is the blade tip clearance drag torque,

T_b is the bearing drag torque,

T_h is the hub fluid drag torque,

T_{disk} is the hub disc friction drag.

The hydraulic driving torque, the hydraulic drag torque and blade tip clearance drag torque can be calculated by numerical simulation. Compared with other torque, the three torque. T_b, T_h and T_{disk} are very small ,so it can be neglected in the balance equation.

Analysis the fluid in the turbine flow meter, at the beginning of impeller rotating, T_{dr} mainly overcomes the bearing drag torque and hub disk mechanical friction retarding torque, and T_i is very small at that time. With the increasing of rotor's rotational speed, T_i becomes bigger. When the rotor is in the steady operation situation, T_i plays an important position while T_b 、 T_h and T_{disk} are neglected. The balance equation can be simplified as

$$T_{dr} = T_i + T_t \tag{4}$$

Governing equation and Simulation methods

Governing equation

From the analysis, we have known that in order to predict the turbine factor, we should known the rotational speed at given flow rate, at this time the torque on the impeller should be balanced. At the beginning, we give a relatively small rotational speed than the balance state. At the given speed and given flow rate we calculate the steady flow field, and the torque on the impeller can't balanced, then in the UDFs we increase the rotational speed gradually, until the torques on the impeller get balanced. This means that the torque on the impeller showed small than 10^{-4}. During this process, momentum equation of the rotational system is depicted as follows:

$$M = J \frac{D\omega}{dt} \tag{5}$$

Where, M is the hydraulic moment vector on the impeller, J is the rotational inertia of the rotational system, ω is angular velocity vector.

As for water in the impeller, the governing equations are still Reynolds-averaged continuity and Navier-Stokes equations in the rotational relative coordinate. While there is still something different, the coordinate is being at an accelerated status. Through deduction, the governing equations are as follows:

$$\nabla \cdot \boldsymbol{W} = 0 \tag{6}$$

$$\frac{\mathrm{D}\boldsymbol{W}}{\mathrm{d}t} = \boldsymbol{f} + \omega^2 \boldsymbol{r} - 2(\boldsymbol{\omega} \times \boldsymbol{W}) - \nabla p / \rho + \nu \Delta \boldsymbol{W} - \frac{\mathrm{D}\boldsymbol{\omega}}{\mathrm{d}t} \times \boldsymbol{R} \tag{7}$$

Where, W is absolute velocity vector in the accelerated rotational relative coordinate, that is relative velocity vector in the Cartesian coordinate;

f is body force vector;

ν is kinetic viscosity coefficient of water;

r is radius vector from the rotational axis to the mass particle;

and R is vector from the origin to the mass particle.

From equation (7), it is seen that there is an additional source force $-\frac{\mathrm{D}\boldsymbol{\omega}}{\mathrm{d}t}\times\boldsymbol{R}$, which is decomposed into three components in directions of x, y, z in the Cartesian coordinate, which are $y\frac{\mathrm{D}\boldsymbol{\omega}}{\mathrm{d}t}$, $-x\frac{\mathrm{D}\boldsymbol{\omega}}{\mathrm{d}t}$ and 0 respectively.

RNG $k-\varepsilon$ turbulence model

Two-equation turbulence models are very widely used, as they offer a good compromise between numerical effort and computational accuracy. The most common two-equation turbulence models are standard $k-\varepsilon$ 、Realizable $k-\varepsilon$ and renormalization group (RNG) $k-\varepsilon$ .In the paper we used the RNG $k-\varepsilon$ turbulence model in order to predicting the turbine meter, which is the most suitable turbulence model for simulation the turbine's vortex and separation (Chen Qingguang, Xu Zhong and Zhang Yongjian, 2003).

The RNG $k-\varepsilon$ turbulence model is similar with the standard $k-\varepsilon$ turbulence model, the k equation and ε transport equation are described as follows (Lam S H. 1992):

$$\rho\frac{\mathrm{D}k}{\mathrm{D}t}=\frac{\partial}{\partial x_j}\left(\alpha_k\mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M \quad (8)$$

$$\rho\frac{\mathrm{D}\varepsilon}{\mathrm{D}t}=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon\mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b) \quad (9)$$
$$-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R$$

The coefficient in the function is deduced from the RNG theory and as follows:

$$C_\mu=0.085, C_{1\varepsilon}=1.42, C_{2\varepsilon}=1.68$$
$$\alpha_k=1.393, \alpha_\varepsilon=1.393$$

Simulation Method

Based on the governing equations in the accelerated rotational relative coordinate, we can calculate the flow field in the turbine meter. The numerical simulation is a unsteady turbulent, as it is in the accelerated rotational relative coordinate. That means that in the CFD process, we should consider the following two aspects to realize simulation of the unsteady transient: 1. Rotational speed of the impeller increases along with time based on equation (7). 2. Additional source forces should be added during the simulation process. According to the above consideration, UDFs (User Defined Functions) is made in Fluent 6.3, and the simulation process is depicted as Fig.2. Steady result at rotational speed n=100 rpm is the initial simulation condition.

The governing equations are discredited with Finite-Volume-Method (FVM). In the unsteady simulation, "Segregated solver", "implicit formulation" and "unsteady time" method are adopted; Second-order implicit format for time item; second-order central difference format for source and diffusion item, second-order upwind format for convection item, and SIMPLEC method is used for velocity-pressure coupling solution. Standard wall function is adopted at the wall contacted with fluid.

Meshed model and Boundary condition

In the simulation process, when the impeller is in rotating, its upstream and downstream flow conditioners are stationary. So the computational domain is composed of moving zone and stationary zone. And the entire turbine meter is divided into three parts, which consist of the upstream conditioners, impeller and downstream conditioners, between the three parts are the boundary "Interface", which can separate the two neighbor flow regions and transfer the vector between the two parts. The data of flow field can be transferred from one region to the other at the interface boundary efficiently.

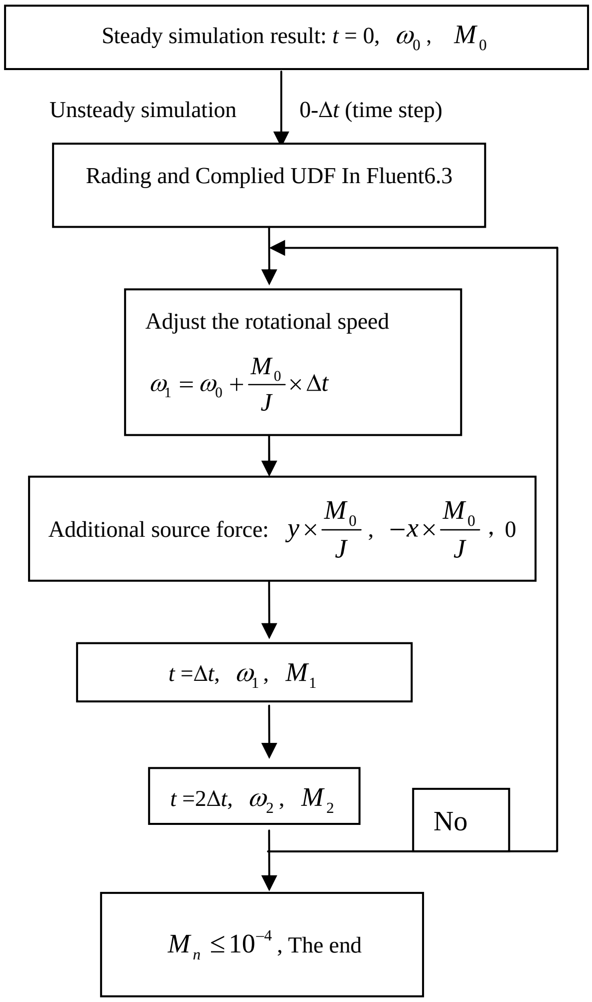

Fig.2 Unsteady simulation process

In the study, the turbine meter is a helix type impeller (Fig. 3), whose outer diameter is 25 mm. The length of the turbine meter, including the upstream conditioner and downstream conditioner(Fig.4), is 138 mm, and the pipe's inner diameter is 26mm. The function of the conditioner is to make the fluid flow into the impeller smoothly, in order to avoid the vortex and other phenomena which could decrease the precision of measuring the flow rate. Meshed turbine flow meter model is shown in Fig.5. For its high complexity of the computation region, it is plot with unstructured tetrahedron mesh of high applicability. Total number of computational meshes is about 0.96 million. In order to get the uniform velocity, 10D upstream pipe and 10D downstream pipe are added at the inlet and the outlet of meter respectively. The fluid medium of numerical computation is water at 20℃, which density and viscosity are 998.2 kg/m^3 and 0.001003 kg/ms respectively.

Inlet condition: mass flow rate, turbulent kinetic energy and turbulent dissipation rate;

Outlet condition: outflow;

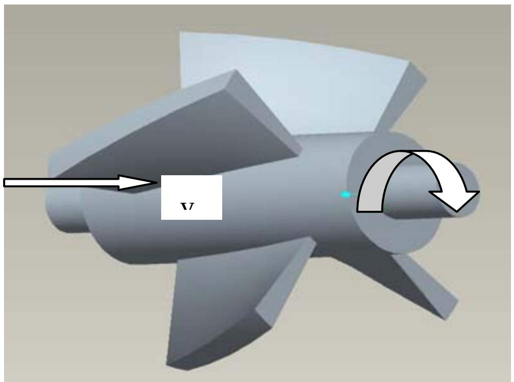

Fig.3 Helix type impeller

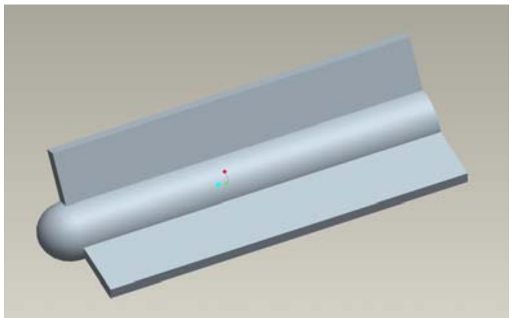

Fig. 4 Conditioner of the turbine flow meter

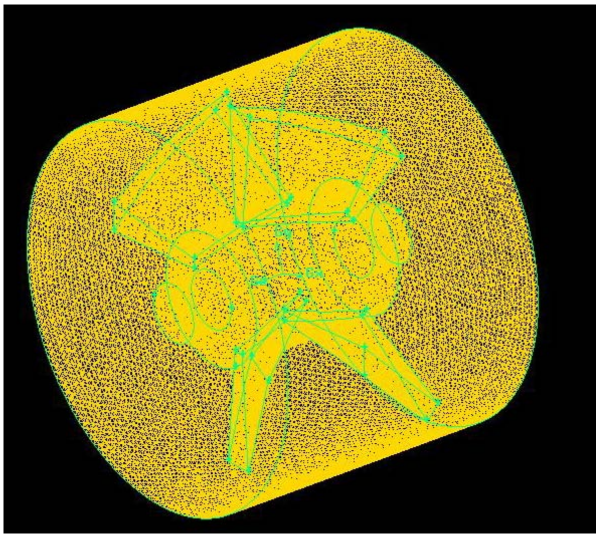

Fig.5 Meshed the impeller

Boundary condition: no-slip condition for the wall, and standard wall function for region near the wall; Steady simulation result under rated working condition as initial flow field of the unsteady simulation.

Numerical simulation results

From the analysis of above, we know that in order to get the meter factor Kc from the numerical simulation, we should know the rotational speed of impeller at the given flow rate, that is the rotational speed at the impeller's torque balance. Through the CFD, we got the meter factor Kc and listed them in table 1. From the data, we get the idea that the turbine meter's meter factor keeps constant at different flow rate, That is the same with the meter factor Ke from the traditional experiment methods. And the errors between CFD and experiment will be discussed in the latter section.

The total pressure distribution on the pressure side and suction side of impeller is plotted in the Fig.6 at the flow rate of $Q=3m^3/h$ at the balance rotational speed, which showed that the total pressure on the pressure side is larger than the suction side. On the front of the suction side which is the largest pressure means that the velocity of this point is zero. And not only on the suction side, but also on the pressure side, the pressure increase from the front edge to the back edge, which means the pressure increases by the impeller.

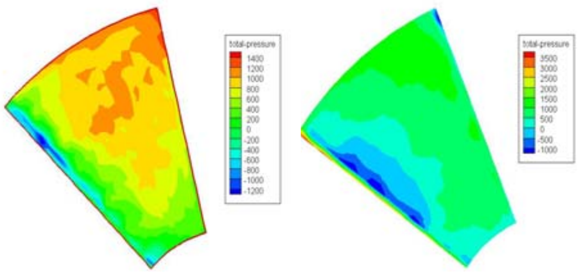

Fig.6 Total pressure distribution on the pressure and suction side

And also the total pressure and velocity distribution at surface of Z=0mm is showed in Fig.7 and Fig.8. The unit of pressure contours is Pa, and the unit of velocity vectors is m/s. From the two figures we know that even at the balance rotational speed, the distribution is in disorder. And the flow field in the turbine meter needs investigate more deeply in order to study the effects of structure on the measuring flow rate's precision. And the development of flow filed with the rotational speed increase also need more research.

Calibration Experiment Research

The calibration experiment has been implemented on the standard flow calibration facilities shown in Fig.9.

Piston type Pipe prover (GB/T 13282-1991) is a standard system used to calibrate the turbine flow meter. Through the equipment the turbine's meter factor can be calculated because the fluid volume flow through the pipe

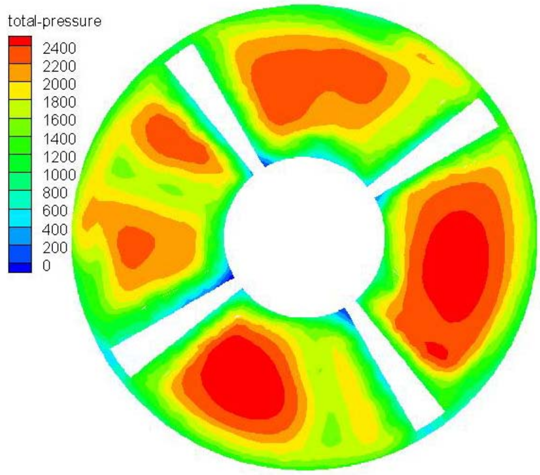

Fig.7 Total pressure distribution at surface of Z=0mm

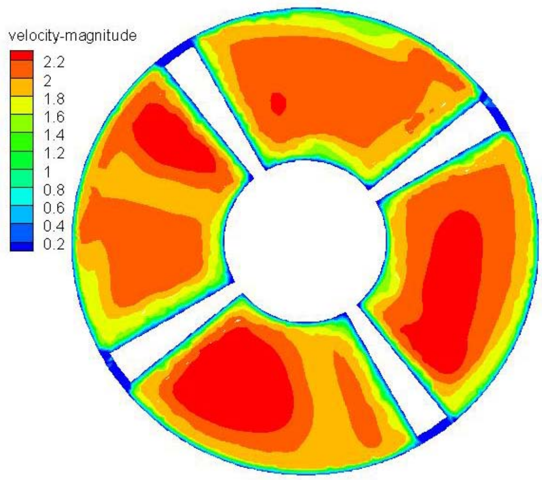

Fig. 8 Velocity distribution at surface of Z=0mm

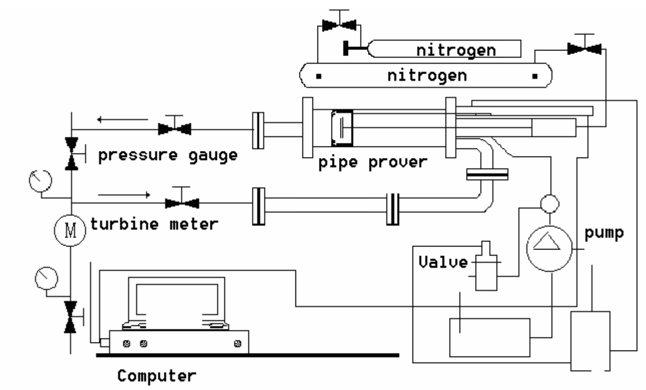

Fig.9 Pipe prover system in calibrate the turbine flow meter

prover is constant. In the calibration process, the computer records the constant volume from the piston type pipe prover and the number of pulse from the turbine meter at the same time at a given flow rate. The turbine's meter factor can calculate by the experiment. The calibration system in the experiment is a system authenticated by National Institute of metrology. P.R. China. Its precision is 0.0292% in calibration a turbine type flow meter. In order to improve the accuracy of the test, three times of measurement were carried out consequently at one test point. According to the standard of JB/T 9246-1999, the meter factor Ke of every test point is calculated, and the values are listed in Tab. 1.

Comparison between CFD and Experiment Research

Comparing simulation meter factor Kc with calibration experiment meter factor Ke, the error between them is also listed in Tab.1, which is calculated by the following formula:

$$Err = \frac{K_c - K_e}{K_e} \times 100\% \qquad (10)$$

According to Tab. 1, the maximum relative error between simulative meter factor Kc and experimental meter factor Ke is about 8.91%, which shows the method finding the meter factor through numerical simulation model is feasible, but it need more accurate in prediction the meter factor considering the bearing friction drag.

Fig.10 showed the meter factor comparison between the calibration experiment and CFD simulation. They showed the same trends, which it is constant when the flow is in fully developed turbulent flow, but at the low flow rate, such as Q=0.4m^3/h, the meter factor showed lager than at other flow rate. It showed non-linear character of the meter factor, which indicates that the bearing friction drag play a different role at different flow rate, and also at the lowest flow rate, the flow in the pipeline is in the turning point between laminar flow and turbulent flow. It is the same with the traditional analysis (Backer 1993).

CONCLUSIONS

In the paper, we introduced a method which can be used to predict the meter factor. The methods, which based on the theory of torque balance on impeller, is quite different from the traditional method in validation the meter factor by experiment. From the paper, two major conclusions are as follows:

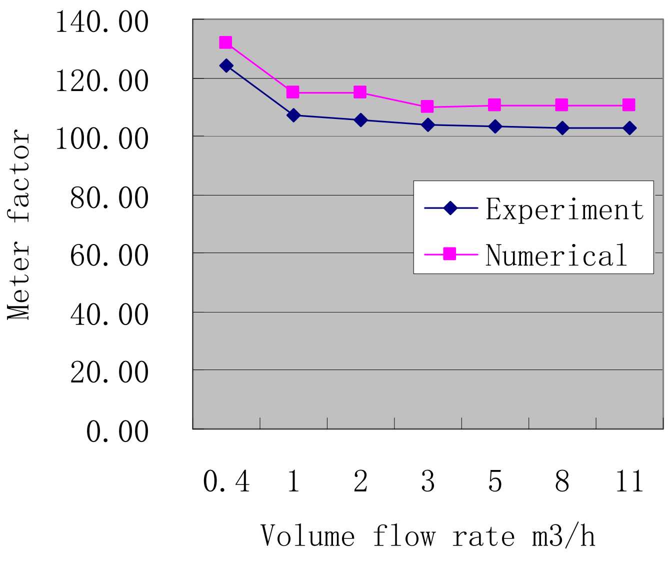

Fig.10 Meter factor comparison between experiment and CFD

1. The methods on the basis of turbulence models and Fluent Code are feasible, and the precision should be improved in the future by considering the bearing friction drag. Comparing simulation meter factor K_S with experimental meter factor Ke, we found the simulative results are consistent with the experimental measurements. The maximum error is about 8.91%, and the trends are the same with the experiments.

2. The flow field of turbine meter is investigated by the rotational acceleration relative coordinate system. In order to improve the accuracy of measuring the flow filed and exploit the application in multi-phase measurement, it is necessary to study the flow field more clearly.

Turbine meter is quite different from the pump and power turbine, the meter aims to cut through the fluid without disturbing the flow. But the unsteady flow conditions can effect its precision, such as the swirl flow in the pipeline. The method used in the paper can used as a useful tool to study the turbine meter in other area, such as the flow rate measuring in multiphase flow. Cavitations can occur in the turbine meter which need study more deeply.

REFERENCES

A.F. Skea , A.W.R. Hall, Effects of water in oil and oil in water on single-phase flowmeters, Flow Measurement and Instrumentation,10(1999) 151-157.

Chen Qingguang, Xu Zhong and Zhang Yongjian, Application of RNG $k-\varepsilon$ models in numerical simulations of Engineering Turbulent flows, Chinese

Quarterly Mechanics, Vol. 24, No.1, 2003 (in Chinese).

Flow measurement calibration facility-Pipe prover, GB/T 13282-1991 (In Chinese)

Lam S H. On the RNG Theory of Turbulence. Physics of Fluids, 1992, 4: 1007-1017.

Qin Hongxin, Chen Kangmin, Zhao Xueduan, Analysis the flow field of impeller of turbine flow meter by FEA method: part1 and part2,Journal of Astronautic Metrology and Measurement,1990, No.6 and 1991, No.2. (in Chinese)

R.C.Baker, Turbine Flowmeters: Theoretical and Experimental published Information, Flow measurement and Instrumentation. 4(1993) 123-144.

Schmidts M, Marliani G, Vasanta Ram VI (1998) A study of the metering error of turbine flow meters caused by swirl in the flow. FLOMEKO 98, Proceedings:399－403

Tomas Humener, E. Von lavante, E.Ernst , Schieber W M, Numerical investigation and design optimization of a 2-Stage turbine flow meter, In: 31st AIAA Fluid Dynamics Conference & Exhibit,11-14 June 2001,Anaheim，CA，AIAA2001-2928.

Turbine flow meter, JB/T 9246-1999, Publishing Company of Mechanical Industrial Press (In Chinese)

Thompson, R.E. and Grey, J. Turbine flowmeter performance model. J Basic Eng., Trans. ASME December (1970) 712-723

Y. Xu, A model for the prediction of turbine flowmeter performance, Flow. Meas. Instrum. Vol.3 No.1, 1992, p37-43

Zheng Dandan, Zhang Tao, Simulation research on turbine meters. Automation and instrumentation, 2005, vol.7, p29-33, (in Chinese)

ACKNOWLEDGEMENTS

The program was supported by a grant from the The National Natural Science Foundation of China (No. 10532010) .The author would like to thank Senior Engineer Zhang JinLin from Tianjin Xinke Whole Set Instrument Meter Co., Ltd, who gives the opportunity to test the turbine meter on the pipe prover system.

Tab. 1 Comparison between numerical result Kc and experimental results Ke

Volume flow rate Q (m^3/h)	Error in calibration %	Meter Factor K_e from Experimental (rpm/L)	Meter Factor K_c From CFD (rpm/L)	Impeller rotary speed n (rpm)	Relative errors (%)
11	-0.3721	102.57	110.25	4701.40	7.48
8	0.0687	102.87	110.32	3429.00	7.24
5	-0.01178	103.26	110.45	2151.20	6.97
3	0.3027	103.69	111.00	1296.16	7.05
2	-0.3454	105.46	114.85	878.82	8.90
1	0.2864	107.90	114.79	449.58	6.39
0.4	0.1489	124.20	131.89	206.99	6.19