CFD MODELING OF TURBINE FLOW METERS

Iryna Gryshanova / Ph.D, Ivan V. Korobko / Ph.D
National Technical University of Ukraine “KPI”

ABSTRACT: *Computational fluid dynamics (cfd) techniques provide investigations in conditions where the real experiment can't be fulfilled for some reason, so these tools have found their applications in many spheres of science and technology; in particular they are widely used in flow metering.*

Some of CFD applications we would like to propose and discuss in this work in the context of turbine flow meters. We discover non-drag type of turbine flow meters to check if created design with hydro-dynamic bearings should provide really floating rotor. For this purpose only numerical research can solve verification problem with minimal costs and simple realisation.

CFD models are used to examine pressure contours and velocity profiles near floating rotor in turbine flow meters. A full three-dimensional flow simulation of realistic meter configurations has been carried out. Close attention was paid to simulation of the complete geometry of turbine meters including front-end and tail-end shapes of flow conditioner hub, metering cells, etc. Some advices to meters' designing were given.

INTRODUCTION

Research work in flow metrology traditionally has deal with a lot of experiments. Currently due to the development of computational fluid dynamics (CFD) researchers have got a chance to substitute expensive full-scale experiments by numerical ones.

Really, CFD is a well-proven technique that is used to visualize and optimize many measuring processes. It is good enough when experimental tests can’t be done for some reasons.

Flow metering is one of primary fields, where CFD tools effectively help to solve such problems as prediction of errors, investigation and modelling of different hydrodynamic effects inside measuring ducts, including visualization of velocity profiles, temperature fields, pressure contours, etc.

CFD numerical simulation is applied for solving Navier-Stokes equations and thus obtaining a velocity distribution law inside metering cell. Its advantages are obvious: low costs with comparatively high computational speed, and ability to predict results in real experiments.

Basically, overview of the numerical tools applications in flow metering gives us a following list of typically solved issues:

- flow meter reaction on flow disturbances such as contractions, expansions, bends, pumps, valves, etc.;
- revealing and correction of the shortcomings in the traditional design;
- investigation of different hydrodynamic phenomenon (vortices, swirls, etc.) in flow measuring ducts.

Today a lot of computational studies are devoted to investigations of turbine flow meters. Searching for the optimal meter geometry has become the fundamental task for their designers. Developing this subject we would like to discuss our vision of CFD modeling while investigating mentioned flow meters.

ANALYSIS ON THEORY OF TURBINE FLOW METERS

Turbine (Woltman) flow meters are frequently installed in water distribution networks. These meters are used for measurement of the water quantity and flow rate in district metering areas.

The primary element of this meter is a turbine (rotor) whose rotational speed ω is directly proportional to the volumetric flow rate.

The angular velocity of the turbine is a function of the driving torque $M_{drive,}$ which is generated by the main stream. This driving torque depends on the velocity distribution and also turbine geometry.

The turbine flow meter operating principle is based on the balance of torques acting on its rotor:

$$I\frac{d\omega}{dt} = M_{drive} - \sum M_{drag}, \qquad (1)$$

where I is the total dynamic moment of rotor inertia, which includes the moment of inertia of the attached flow weight, dω is the change of angular speed ω of turbine rotation in time t, M_{drive} is the driving torque acting on the turbine, $\sum M_{drag}$ is the sum of all drag torques acting on the rotor.

The major source of error for these flow meters is in friction which takes place between the hub of the turbine and the bearing hub and between the turbine blade tips and the flow meter housing.

The error due to bearing friction can be minimised by application of hydro-dynamic bearings. For today we know some design principles to create these bearings [1, 2]. But experimental verification of meters with hydrodynamically balancing rotor is pretty difficult task. The key information for its solving should be obtained through analyzing the static pressure field inside the meter. The most reasonable way to do it is based on numerical investigation of an axial type turbine flowmeter with application of CFD code FLUENT.

The aim of the evaluation was to study the efficiency of the created meter geometry for reducing of bearing friction. So, we should obtain the curve of static pressure distribution along the measuring duct and analyze it for presence of ‘floating rotor’ effect.

The shape of the curve depends upon the viscosity of the liquid which varies with temperature, as well as on the flow rates on which the flowmeter is used. Designing turbine flow meters researchers look for optimal tip clearance of the turbine, shape of the rotor blades and the hub/rotor ratio. All parameters somehow affect each other, so the main task consists in selection of correct sizes and their relations inside flow measurement section. It may take a lot of time if we will find rational size relations in usual experimental tests so computational study in searching for optimal turbine meter design would be much appreciated.

COMPUTATIONAL STUDY OF THE TURBINE FLOW METERS WITH FLOATING ROTOR

DN 50 mm turbine meter's flow fields were simulated with CFD software FLUENT. We created some designs of turbine and conditioners (see fig.1 and fig.2) to provide a few numerical experiments.

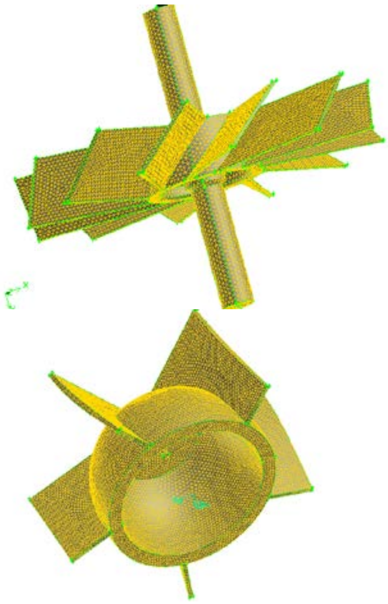

Fig. 1 Meshed models of turbine and conditioner created in GAMBIT

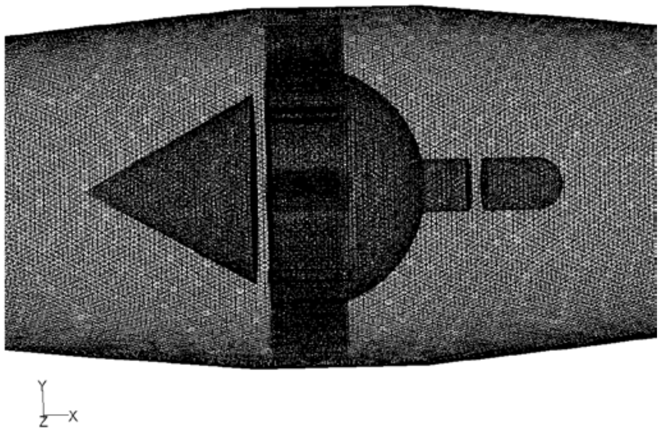

Fig. 2 Meshed model of floating rotor created in GAMBIT

A CFD simulation has been carried out to help in determination of pressure distribution inside flow measurement section to find out the best position of the rotor to achieve the hydraulic balance point.

Numerical results

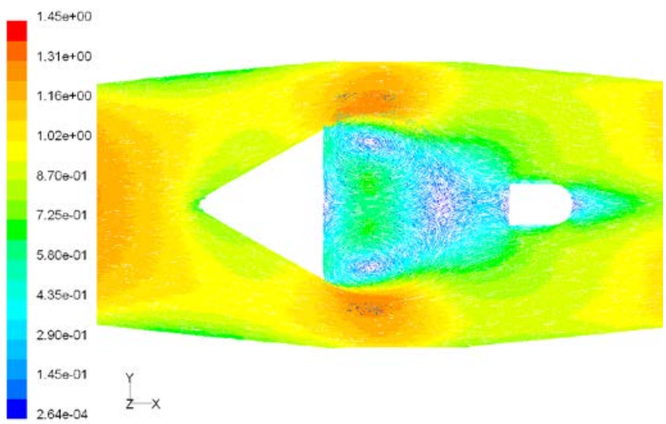

Fig. 3 Velocity vectors colored by velocity magnitude for 'floating rotor' while Q=10 m^3/h

Flow fields show that meter's pressure change is influenced by front-end and tail-end conditioner shapes, radius of flow conditioner hub, the thickness of rotor blades.

There are vortices in gaps in front and back of rotor hub. Conditioner affects the flow field around the rotor. The rotation of rotor influences on flow fields closely in front and beyond of it. So, having numerical model we can visualize the hydrodynamics inside meter and change it for achieving our goals.

To prove the effect of lowering the rotor bearing friction it was enough to plot static pressure distribution curves along metering cell at different flow rates.

As far as modelling was conducted at different flow rates so fixed boundary conditions were as following:

- for flow rate Q = 0,45 m^3/h
  inlet velocity V= 0,06 m/s, rotational frequency n=1,88 rad/s ;
- for flow rate Q = 10 m^3/h
  inlet velocity V= 1,42 m/s, rotational frequency n=50,24 rad/s ;
- for flow rate Q = 30 m^3/h
  inlet velocity V = 4,246 m/s, rotational frequency n=151,98 rad/s .

All plots (fig. 4, 5, 6) demonstrate the same phenomenon, which consists in pressure lowering around the rotor and its increasing behind the rotor. In other words, zones with increased pressure located before and beyond the turbine do not allow it to move forward and back under flowing liquid. So the turbine will be kept in balance.

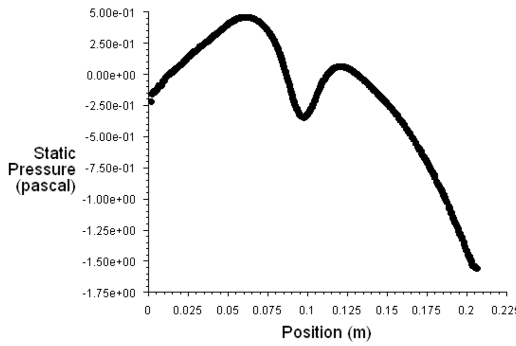

Fig. 4 Static pressure distribution curve along metering cell while Q=0,45 m^3/h

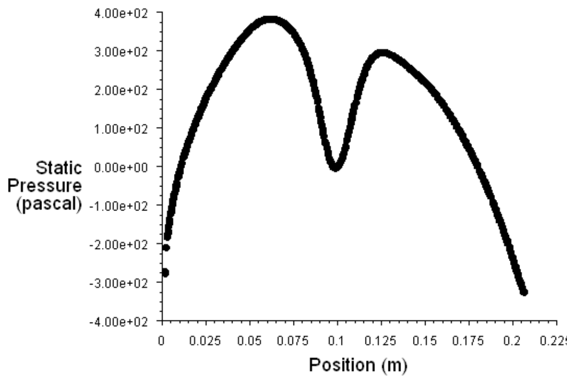

Fig. 5 Static pressure distribution curve along metering cell while Q=10 m^3/h

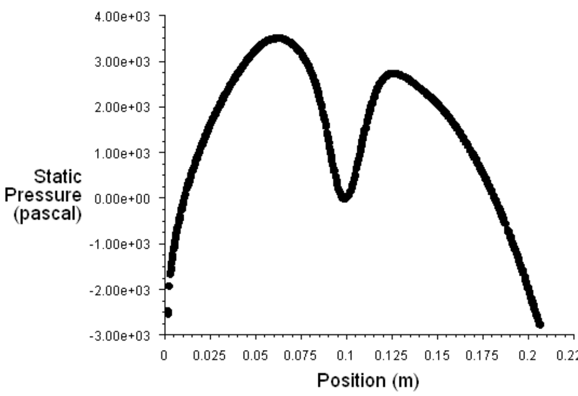

Fig. 6 Static pressure distribution curve along metering cell while Q=30,0 m^3/h

The results show that the simulation model is effective and can be used not only for selection of rational sizes of measuring duct. It is also good for examining created design on ability to provide a hydraulic balance mechanism for rotor in all points of measuring range and so to protect axial bearing from excessive deterioration.

CONCLUSIONS

The main task of CFD simulations in flow metering is to give effective tool for calculating the measurement uncertainty. Based on these techniques we discovered a possibility to improve performance of turbine flow meters. During this study we've reached the following conclusions:

1. This work confirmed CFD usefulness as a design tool. Even reduced accuracy simulation proved useful because we could obtain calibration curves which coincide with real calibration curves practically in all measurement range except for transition zone, which desirably should be avoidable.
2. Designing turbine flow meters we could apply CFD to obtain pressure distribution in measuring duct and thereby to check the effect of hydrodynamic balance for rotating turbine. This effect significantly increases performance and longevity of turbine meters though its proving in real conditions is complex and expensive task.
3. With the continual improvement of grid generation 3D modelling of a flow field should soon be more practical for many flow metering problems.

REFERENCES

[1] Ing C. J. Benard, 1988, 'Handbook of fluid flowmetering', Trade & Technical, pp. 29-30.

[2] Kremlevsky P.P., 1989, "Flow meters and quantity meters", *Energy Publ. Co.*, Leningrad, Russia.